Titre: Title:	Integrated numerical methods for optimization of gear trains
Auteur: Author:	Jun Yang
Date:	2007
Type:	Mémoire ou thèse / Dissertation or Thesis
Référence: Citation:	Yang, J. (2007). Integrated numerical methods for optimization of gear trains [Mémoire de maîtrise, École Polytechnique de Montréal]. PolyPublie. https://publications.polymtl.ca/8032/

URL de PolyPublie: PolyPublie URL:	https://publications.polymtl.ca/8032/
Directeurs de recherche: Advisors:	
Programme: Program:	Non spécifié

UNIVERSITÉ DE MONTRÉAL

INTEGRATED NUMERICAL METHODS FOR OPTIMIZATION OF GEAR TRAINS

JUN YANG

DÉPARTEMENT DE GÉNIE MÉCANIQUE
ÉCOLE POLYTECHNIQUE DE MONTRÉAL

MÉMOIRE PRÉSENTÉ EN VUE DE L'OBTENTION
DU DIPLÔME DE MAÎTRISE ÈS SCIENCES APPLIQUÉES
(GÉNIE MÉCANIQUE)

MAI 2007

Your file Votre référence
ISBN: 978-0-494-35705-7
Our file Notre référence
ISBN: 978-0-494-35705-7

UNIVERSITÉ DE MONTRÉAL

ÉCOLE POLYTECHNIQUE DE MONTRÉAL

Ce mémoire intitulé :

INTEGRATED NUMERICAL METHODS FOR OPTIMIZATION OF GEAR TRAINS

présenté par : YANG, Jun

en vue de l'obtention du diplôme de : Maîtrise ès sciences appliquées

a été dûment accepté par le jury d'examen constitué de :

M. LÉVESQUE, Martin, Ph. D., président

M. VADEAN, Aurelian, Doct., membre et directeur de recherche

M. SANSCHAGRIN, Bernard, D.Ing, membre

ACKNOWLEDGEMENTS

I wish to express special thanks to my thesis director, Professor Aurelian VADEAN. He has given me great academic support and valuable advice during the entire research process. Without his patient directing and kindly encouragement, the goals of this thesis would not have been accomplished successfully.

Thanks must be given to CRIAQ, who provided me with such a good research project as well as financial support.

I would like to thank the specialists who formed our partnership for this project. They are: Mr. David Venditti, Mr. Benoit Blondin, and Mr. Martin Poulin of Pratt & Whitney Canada; Prof. Chevy Chen of Concordia University; and Mr. Pablo Vasquez of ROI Engineering. They have all given me much help and many good suggestions.

Finally, and respectfully, I would like to express my gratitude to my parents, my wife, Yanyu, and our lovely daughter, Hélène, for their understanding and love, as well as providing material and spiritual support during my studies.

RÉSUMÉ

Cette thèse a développé une procédure d'analyse et d'optimisation par éléments finis spécialement pour des trains épicycloïdaux, en intégrant différents outils d'aide à la conception.

L'utilisation extensive des grands outils commerciaux est considérée comme une méthode efficace pour réduire les coûts de développement et améliorer la qualité des produits. Bien que les outils commerciaux soient généralement capables de traiter les structures mécaniques, il n'est pas garanti qu'ils puissent analyser automatiquement des systèmes complexes tels que les trains épicycloïdaux. Les études menées dans ce travail ont relevé tous les problèmes rencontrés lors de l'analyse par éléments finis et l'optimisation des trains épicycloïdaux. Un groupe de solutions est proposé pour régler les problèmes dans chaque étape.

Afin d'obtenir une procédure d'optimisation plus facile à contrôler et plus flexible, un optimiseur basé sur les algorithmes génétiques est programmé et appliqué à la conception des systèmes d'engrenages. Bien que "DesignXplorer" dans "ANSYS Workbench" est un module d'investigation très puissant, sa routine assez rigide ne permet pas aux concepteurs d'intervenir lors de l'optimisation. On peut pourtant obtenir des informations très intéressantes quant à l'influence des différents paramètres de conception et les utiliser pour construire la fonction objective d'un optimiseur externe. L'application des algorithmes génétiques constitue une méthode d'optimisation stochastique qui convient pour traiter presque toutes les optimisations des systèmes. L'optimiseur basé sur les algorithmes génétiques est validé par deux modèles dont un modèle analytique et une modélisation par éléments finis qui sont issus d'un même train d'engrenages. Le processus d'optimisation et les résultats ont prouvé que cet optimiseur peut être appliqué à l'optimisation des trains d'engrenages. En l'intégrant avec des outils d'analyses commerciaux, l'optimiseur est capable de réaliser une conception optimale automatique.

ABSTRACT

This thesis has developed a finite element analysis (FEA) procedure and an optimization procedure for the epicyclic gear system, by integrated with a powerful and comprehensive computational tool – ANSYS Workbench CAE environment.

The FEA method has been studied as a robust analysis tool for the design and analysis of complex gear systems. Extensive use of large commercial tools is recognized as an efficient way to reduce subsequent development costs and improve product quality. Although commercial tools are capable of processing most mechanical structures, it is not guaranteed that they will automatically analyze complex systems, such as the epicyclic gear system. The studies in this thesis reveal all the problems encountered while employing finite element analysis and optimization on epicyclic gear trains; and a set of solutions to surmount these problems is provided in each step.

In order to attain a more controllable and flexible optimization procedure, Genetic Algorithm optimizers were programmed and applied to the optimization design of mechanical gear system. Although ANSYS Workbench has its own powerful optimization module – DesignXplorer, the fixed routine makes it difficult for mechanical designers to make desired changes on optimization. We can obtain very useful information – the influence of different design parameters on the output parameters – and use this information to build the objective function in the external optimizer.

Genetic Algorithm is a derivative-free stochastic optimization method that is suitable for dealing with almost all kinds of system optimization. This GA optimizer has been developed and validated on both the analytical model and the finite element model of the same gear system. The optimization procedures and their results have proven that this GA optimizer can be directly applied to the FEA- based optimal design of complex gear systems. Integrated with commercial analysis tools, this optimizer is able to realize a highly automated optimization design.

CONDENSÉ EN FRANÇAIS

CHAPITRE 1 INTRODUCTION

Les systèmes mécaniques dans l'industrie aérospatiale sont essentiels pour la sûreté et la longévité des avions. En raison de leurs applications spécifiques, les produits aérospatiaux doivent posséder la qualité et la fiabilité requise avec un poids minimum.

Une des difficultés rencontrées dans le cycle de développement des produits est que les divers outils de conception et d'analyse emploient différents formats de fichier créant souvent des problèmes en échangeant des données d'un à l'autre. Afin de pouvoir réduire ce cycle ou d'éliminer certaines étapes chronophages nous avons proposé la méthodologie suivante :

- Optimisation et validation basées sur les résultats obtenus lors des analyses par éléments finis :

  L'environnement d'ANSYS est choisi avec ses modules avancés Design Modeler, Simulation et DesignXplorer. Puisque tous ces modules sont dans l'environnement d'ANSYS, il n'y a ni difficulté de communication ni conflit dans la transmission des fichiers. Tout le processus est capable d'être suivi automatiquement.

- Algorithme génétique sur l'optimisation du produit :

  Pour l'optimisation de la conception du produit, cette recherche propose une nouvelle approche qui emploie un algorithme génétique (GA) pour trouver le minimum/maximum global. La complexité, la fiabilité et la supériorité de l'approche seront analysées et comparées aux méthodes conventionnelles. La conception des trains d'engrenages servira de repère pour valider la nouvelle méthodologie dans la conception de produit.

CHAPITRE 2 REVUE DE LA LITTÉRATURE

Ce chapitre est constitué de quatre sections. La première section se concentre sur les méthodes d'analyse de contraintes et de déformations des trains d'engrenages. Une des méthodes importantes discutées ici est la méthode d'analyse par éléments finis. Dans la deuxième section, les travaux différents d'optimisation sur les trains d'engrenages seront présentés. La méthode traditionnelle d'optimisation ainsi que la méthode d'optimisation par algorithmes génétiques sont détaillées et comparées. La troisième section montre les études précédentes sur l'optimisation multi objectifs. La méthode d'optimisation multi objectifs classiques et les algorithmes évolutionnaires seront présentés. Enfin, le développement de l'intégration des outils d'analyse et d'optimisation sera discuté dans la dernière section.

CHAPITRE 3 ANALYSE PAR ÉLÉMENTS FINIS DES TRAINS D'ENGRENAGE ÉPICYCLO ÏDAUX

Ce chapitre présente une procédure d'analyse par éléments finis appliquée aux trains d'engrenages épicycloïdaux. Le fonctionnement d'un train épicycloïdal présente de nombreux contacts avec glissement. Les contacts avec glissement imposent une analyse non linéaire et par la procédure itérative nécessaire le temps de calcul est augmenté. En outre, une définition de contact non précise aura comme conséquence la non convergence de la solution. Ainsi, pour éviter ce genre de situation et afin de proposer un procédé d'analyse robuste, la simplification du train d'engrenages est exigée dans l'étape préliminaire de la procédure.

L'analyse par éléments finis est appliquée sur le modèle simplifié du train d'engrenages. Le modèle a été établi dans "CATIA V5" et importé dans "ANSYS Workbench". Par la suite, plusieurs opérations ont été effectuées :

- Réduire le modèle global à un tiers ;
- Réaliser les maillages et appliquer les conditions limites ;

- Exécuter la simulation et post-traiter les résultats.
- Effectuer une étude des convergences des résultats.

Ce procédé convient à l'analyse par éléments finis d'un train d'engrenages épicycloïdal. Bien que ce procédé soit utilisé sur un train d'engrenages simplifié, le traitement d'un train d'engrenages épicycloïdal complet pourrait être également accompli avec seulement une préparation supplémentaire pour la réalisation du maillage et l'application des conditions aux limites.

CHAPITRE 4 L'OPTIMISATION DU TRAIN D'ENGRENAGES ÉPICYCLOÏDAL DANS "ANSYS WORKBENCH"

Puisqu'une conception préliminaire peut être très complexe et posséder beaucoup de caractéristiques spéciales, une méthode d'optimisation basée sur une démarche analytique est difficile à réaliser. Le grand nombre de paramètres de conception d'un système augmente également les difficultés d'optimisation du système global. Un procédé d'optimisation du modèle géométrique en utilisant un outil commercial puissant peut fournir un résultat optimal complet et satisfaisant. Ce chapitre présente la méthodologie pour exécuter l'optimisation sur le train d'engrenages épicycloïdal dans l'environnement d'"ANSYS Workbench".

DOE (Design of Experiments) est une technique de recherche générale adoptée par la méthode d'analyse par éléments finis pour déterminer le choix des configurations de conception. Des points de prélèvement sont localisés tels que l'espace des paramètres d'entrée soit exploré de la manière la plus efficace. Avec la préparation appropriée dans le module de simulation, l'exécution d'une optimisation DOE est moyennement complexe. Avant l'optimisation, tous les paramètres d'entrée et les paramètres de sorti doivent être exposés et liés convenablement afin d'assurer la mise à jour réussie de la géométrie pendant l'exploration de l'espace de conception. Dès que l'optimisation est achevée, des surfaces de réponse sont obtenues et l'étude de sensibilité peut être réalisée. L'étude de sensibilité fournit une vue globale sur les paramètres de conception ainsi que leur influence, ce qui aident le concepteur à contrôler correctement le procédé d'optimisation.

L'objectif d'optimisation pour le train d'engrenages épicycloïdal aéronautique a été choisi comme une combinaison de la masse globale et des contraintes de von Mises. C'est une optimisation multi objectifs. Le but est de réduire au minimum la masse globale tout en visant l'exploitation de la capacité de charge maximum du matériau, soit des contraintes de von Mises le plus près de la limite supérieure. Dans un modèle de train d'engrenages, trois candidats à l'optimisation sont obtenus. En considérant comme priorité les contraintes devant la masse (trade-off study), nous avons pu obtenir une réduction de 3.8% de la masse totale du train.

CHAPITRE 5 DÉVELOPPEMENT D'UNE MÉTHODE D'OPTIMISATION BASÉE SUR LES ALGORITHMES GÉNÉTIQUES

La deuxième partie de cette recherche est le développement d'un procédé d'optimisation basé sur les algorithmes génétiques. Un optimiseur est développé avec une stratégie générale d'algorithme génétique (Goldberg, 1989). Cet optimiseur GA peut traiter l'optimisation avec un seul objectif et l'optimisation multi objectifs. L'optimisation avec un seul objectif est relativement facile puisque l'objectif d'optimisation est directement traité comme le critère de choix. Mais pour l'optimisation multi objectifs, elle est tout à fait complexe puisque la conception optimale n'est pas un seul point mais un groupe de points d'étude, qui sont issus des buts contradictoires d'optimisation, soit le volume total et la contrainte maximum.

Pour employer une méthode d'optimisation multi objectifs, une fonction objectif est construite en intégrant un logarithme comme fonction de pénalité. Cette fonction équilibre les critères de sélection en recherchant une combinaison des variables de conception pour répondre à toutes les exigences de la fonction objective. Ceci peut être réalisé en ajustant le facteur pondérateur de chaque objectif d'optimisation. L'objectif plus important aura une pondération plus élevée. Par contre, tout critère de sélection n'est pas sensible aux changements de l'objectif avec la pondération la plus basse. À la fin du processus d'optimisation, le meilleur candidat est issu de la combinaison de tous les objectifs d'optimisation.

CHAPITRE 6 DÉVELOPPEMENT DES OPTIMISEURS SPÉCIFIQUES DE GA ET L'INTÉGRATION AVEC LES DIFFÉRENTS OUTILS D'ANALYSE

Dans ce chapitre, deux optimiseurs GA sont développés dans l'environnement C++ et l'environnement MATLAB. Un modèle de train d'engrenages simplifié a été développé pour vérifier le procédé d'optimisation du module de GA. Le modèle est un train d'engrenage avec deux arbres et deux engrenages.

Pour l'optimiseur sur C++, la contrainte de von Mises maximum du modèle est considérée comme un objectif d'optimisation. Un autre objectif est le volume total du train d'engrenages. Une optimisation basée sur une formulation analytique est exécutée. La conception optimale a été trouvée et l'optimiseur GA est validé.

Un autre optimiseur GA est développé en utilisant le langage de programmation MATLAB. L'optimiseur GA en utilisant MATLAB est appliqué sur un modèle analytique et un modèle par éléments finis 3D d'un même train d'engrenages. Le modèle analytique peut fournir la solution préliminaire pour le système mécanique. La méthode d'analyse par éléments finis peut résoudre le problème mécanique complexe avec une solution plus précise, en considérant l'apport de toutes les pièces composant le système.

L'optimiseur GA est évalué sur les deux modèles. Les résultats prouvent que les objectifs d'optimisation sont atteints et toutes les contraintes sont respectées. Le volume total minimum est convergent et l'objectif de la contrainte de flexion sur les dents d'engrenage est stable et prêt de la limite prévue.

CHAPITRE 7 CONCLUSION

Une procédure générale d'analyse par éléments finis sur le train d'engrenages épicycloïdal a été développée lors de ce travail. Les différentes étapes ont été explicitées dans l'environnement d'"ANSYS Workbench". Ensuite, un procédé d'optimisation a été développé afin de trouver la conception optimale pour un train d'engrenages épicycloïdal donné.

Afin d'obtenir un module d'optimisation plus flexible et mieux contrôlable pour le train d'engrenages épicycloïdal, deux optimiseurs ont été développés en utilisant les algorithmes génétiques. Le procédé d'optimisation recherche les « meilleurs » candidats en simulant le procédé de sélection naturelle. Une fonction multi objectifs a été développée pour traiter les variables de sortie contradictoires de conception. La fonction de pénalité a été utilisée pour amener ces variables près des valeurs à atteindre. Les résultats d'optimisation ont été obtenus à partir des deux modules d'optimisation. Les chiffres de convergence prouvent que des résultats optimaux peuvent être atteints d'une manière efficace. L'optimiseur GA a été validé sur un modèle simple d'une transmission de puissance et ensuite testé sur un train d'engrenages.

CONTENTS

LIST OF FIGURES

LIST OF NOTATIONS AND SYMBOLS

F:	Gear Face Width;
k:	the contact stiffness;
Δ :	amount of penetration;
D':	relative deflection of carrier pin;
D_1:	maximum tangential displacement on one end of carrier pin;
D_2:	tangential displacement on the other end of carrier pin;
L:	carrier pin length;
Sen_j :	single parameter sensitivity for design parameter j;
y_{max}:	corresponding upper response parameter value for input parameter j;
y_{min} :	corresponding lower response parameter value for input parameter j;
y_{ini} :	initial response parameter value;
P_j:	sensitivity percentage for design parameter j;
f:	fitness function or objective function;
V:	total volume;
$s_1, s_2, \ldots, s_n$:	optimization objectives, i.e. maximum stresses, maximum deformation;
$S_1, S_2, \ldots, S_N$:	allowable limits of optimization goal for each objective;
$a_1, a_2, \ldots, a_n$:	weighting factors for optimization objectives;
ϕ :	pressure angle;
σ:	maximum bending stress;
τ:	maximum torsional stress;
l:	the length of shaft;
σ_{equiv} :	maximum von Mises stress;
T_1:	input torque;
R_1:	radius of pinion;
R_2:	radius of gear;
r_1:	radius of pinion shaft;
r_2:	radius of gear shaft;

I: moment of inertia;

c: distance from neutral surface to outer surface

LIST OF TABLES

LIST OF ANNEXES

CHAPTER 1

INTRODUCTION

1.1 Context

The mechanical systems used in the aerospace industry are vital for the safety and durability of aircrafts. Due to their special applications, aerospace products need to possess a high quality and reliability with minimum weight. In designing these products, not only should the geometries of all their components be generated, but their functions, attributes, and levels of safety should be evaluated also. To account for all these factors, two design stages are carried out, initial and advanced. In the initial design stage, the components' geometries are generated and a preliminary analysis of their various attributes and functionalities is conducted. In the advanced design stage, detailed analyses are carried out using various computational tools, such as finite element analyses, computational fluid dynamics, heat transfer, manufacturability systems, etc., as necessary. On the basis of these analyses, adjustments are made to the components, and the product functions are compared with those of the initial design in order to optimize the final design. Due to high competition in industry, and technical difficulties in product design, companies are making extensive use of powerful computational tools for design and analysis in order to reduce subsequent development costs and improve product quality. But, while computational tools can help designers have a deeper understanding of the anticipated performance of their products from a different discipline's perspective, achieving the optimum design poses a number of technical challenges which hinder the wide application of computational tools and the attainment of a generic solution to design optimization. This research proposes the development of a practical approach to optimize product design: the design of gear trains, a critical mechanical product, which will be used to illustrate the proposed approach.

1.2 Research Motivation

This research has been proposed by a principal aircraft engine manufacturer. Consider then the case of the accessory gear boxes (AGBs) in aircraft turbine engines. Their gear trains usually consist of several pairs of meshed gears, splines, shafts, carriers, and accessories. In the gear train design process: (1) an in-house gear design system is used to calculate the initial design parameters of the gear train; (2) the components of the gear train with reduced parameters are saved in individual files which are fed manually into the CAD/CAM system to generate the 3D models for each component; (3) assembly of the gear train is generated and finite elements are meshed; (4) the mesh data of each 3D model is imported into the finite element analysis (FEA) program to determine the maximum bending and contact stresses and, subsequently, to modify the initial design of each component based on the analysis results; and (5) if the stresses are beyond specified limits, another design loop is initiated by changing the initial design; otherwise, the design is considered satisfactory and saved as final. This design approach will be examined to identify the technical challenges, and find effective ways to tackle them in order to arrive at an optimized design process.

One of the difficulties encountered in product design is that the various design and analysis tools use different proprietary file formats, often creating problems in the exchange of data between the tools. At an aircraft engine manufacturer, three main design and analysis tools are utilized for gear train design: gear design system, CATIA, and FEA tool, in addition to the auxiliary tools and spreadsheets. In the initial design of the gear train, gear design results are saved in a text format which cannot be read by CATIA; thus it becomes necessary to input them into CATIA manually, a time-consuming and error-prone process. In turn, the CATIA mesh data of each component cannot be loaded directly into the FEA program, so a data converter is required for the data exchange. With this in view, a product-data management method, integrating the computational tools, is proposed in order to develop data exchange problem-solving in a cost-effective way.

Another difficulty associated with the design of a mechanical product involving many design parameters for each of the components, is that of satisfying the minimum weight criteria for the product. In gear train design, the analytical study conducted on the gears, splines and other components does not offer satisfactory weight minimization capabilities. This is because the objective function of the product's weight with respect to all design parameters is impossible to formulate, and conventional optimization approaches cannot solve the design optimization problem. Since, in theory, the design can be optimized by adjusting pressure angle, tooth thickness, root diameter, outside diameter, etc., it is proposed that a non-conventional optimization method be established in order to find the solution for this non-linear, discrete, and constrained problem. A finite element analysis of the final design, with all its features, is an ideal analysis method due to its capability of providing a complete analysis result. Since the finite element analysis is applied directly onto 3D geometry models from CAD systems, formulation problems on the relationship between results and design parameters can be solved by the powerful FEA method.

1.3 Proposed Methodology or Approaches

The proposed methodology is as follows:

- Optimization and Validation based on FEA

  Once the preliminary design is performed and all the geometries are generated, it is necessary to carry out a complete stress analysis in order to validate the design, improve it as necessary, and execute, according to its specific optimization. For this phase of the project, the CAE ANSYS Workbench environment will be chosen, using the advanced modules of Design Modeler (parametric models), Simulation (FEA), and DesignXplorer (Optimization based on Design of

Experiments and/or Variational Technology). Since all these functional modules are in one ANSYS environment, there are no communication difficulties and no format conflicts with file transfer, and all the processes are capable being carried out automatically.

- Genetic Algorithm on Product Design Optimization

  For product design optimization, this research proposes a new approach that uses the Genetic Algorithm (GA) to find the global minimum/maximum within product design optimization.

  GA is a derivative-free stochastic optimization method based on the concepts of natural selection and evolutionary processes (Goldberg, 1989). GA is now well-established as a general-purpose global optimization tool. To find the optimum design of a product, GA is applied to the FEA-based optimization of gear trains. Since both the finite element analysis and optimization tool are quite mature, the proposed product design optimization approach will be stable and quite feasible. The computation complexity, reliability, and superiority of the approach will be analyzed and compared with conventional methods. The design of gear trains will serve as a benchmark to validate this new methodology in product design.

1.4 Thesis Organization

The research work of this thesis is composed of seven chapters. The literature review in Chapter 2 introduces early studies in the scope of finite element analysis, optimization approaches on gear system, and integration of different analysis and optimization tools.

Chapter 3 presents the development of FEA procedures on epicyclic gear train systems. Due to the compact size of the part's geometry and the wide variation range of transmission ratios, the epicyclic gear system is widely used in the aircraft and automobile industries. This chapter illustrates the problems encountered while applying a stress and deformation analysis (FE method) on epicyclic gear systems, and provides corresponding solutions for the problems.

In Chapter 4, an optimization strategy for use on epicyclic gear trains is described while implementing the powerful computational tool – ANSYS Workbench CAE environment. A sensitivity study is performed in order to locate the influence of each design parameter upon the specific output parameters. This study also provides very important information on the selection of design parameters for both preliminary optimization and second-stage refined optimization. Multi-objective optimization is achieved with the Goal Driven Optimization (GDO) method. This is a "trade-off" optimization method employed in order to balance the opposite trends of different design parameters, according to the importance of each parameter.

In Chapter 5, an optimization method is developed based on the Genetic Algorithm (GA). Using nature selection theory, design variables from an optimization model are combined in order to become selection candidates. Analysis of each candidate is performed and the fitness of them is obtained. The candidates with the best fitness are selected as the parents to generate the next generation of candidates. The process is repeated until the "stop" criterion is met. The best candidate from the latest generation is kept as the optimization result.

Multi-objective function is also created to solve the conflicting objective optimization problem. Different objectives and weighting factors are combined to build the multi-objective function. A penalty function is used to push the objectives so that they reach their target value rapidly.

In Chapter 6, two GA optimizers are developed in both the C++ language environment and the MATLAB language environment. C++ language is a powerful programming language, thus it is suitable for building a complex optimization system with graphic user interface. As MATLAB is more concise and flexible it is convenient for engineering computing.

One GA optimizer was developed with C++ programming language. A one-stage gear train model was developed to verify the optimization procedure of the C++ GA module. Another GA optimizer was developed with the same optimization procedure in MATLAB environment. MATLAB provides many useful mathematical functions as well as the management of flexible variables. It is the more suitable choice for scientific research, as in the preliminary development stage of the GA optimizer within the scope of this thesis. The GA optimizer programmed in MATLAB language was validated by a simple one-stage gearbox. Optimization results on both the analytical model and the FEA model were obtained and compared. The results proved that the developed GA optimizer could find the optimal values of design variables. Furthermore, an automatic design system was created by integrating GA optimizer with the ANSYS FEA tool. This integrated design system was able to realize an automatic finite element analysis and optimization on the gear train.

Finally, in Chapter 7, conclusion of this thesis is drawn and prospective research is discussed.

CHAPTER 2

LITERATURE REVIEW

The epicyclic gear train, also called the planetary gear train, is widely used in aircraft power transmission systems. Due to the highly compact size of the part's geometry and a large speed ratio between the input and output shafts, the epicyclic gear train holds a very important role in both the reduction gearbox (RGB) and accessory gearbox (AGB) of different kinds of aircrafts.

Over the last several decades, the gear train's analysis, and optimization have been, perhaps, the two oldest mechanism topics that have attracted many researchers' attention. Before the emergence of the modern, powerful computer and comprehensive structural analysis software, this work was based mostly on the mathematical or analytical formulas method; plotting figures from the experiments data. With the rapid development of the computation capabilities of computers and the gradually-perfected commercial analysis software, the analysis and optimization of this complex mechanical system has become possible, thanks to these powerful tools.

This chapter, focused on the literature, will review the history of the development of analysis and optimization studies on gear systems. This chapter is composed of four sections. The first section will focus on the stress and deformation analysis methods employed on gear trains. One of most important methods discussed here is the finite element analysis method. In the second section, different optimization research work conducted on gear trains will be presented. The traditional optimization method and the genetic algorithm optimization method will be detailed, and compared. The third section will show earlier studies on multi-objective optimization. Both classical multi-objective optimization method and evolutionary algorithms will be introduced. Finally, the developing integration of analysis and optimization tools will be discussed in the last section.

2.1 Finite Element Analysis on Gear Train

During the design and analysis procedures conducted upon gear trains, the accurate prediction of maximum stresses on, and the deformations in, gears requires a feasible and efficient method; a mathematical description, or the finite element analysis method.
A great amount of research work is based on the former method. Liu and Tsay (2001) derived mathematical descriptions of conical involute gear tooth flanks and fillets. But intervals for the surface parameters and addendum modification are not presented in their formulas. To provide a more complete mathematical description of conical involute gears, Jesper Brauer (2004) derived a general finite element model which is based on derived analytical representations of their shape. This description is capable of presenting four types of involute gears: spur gears, helical gears, straight conical gears and conical involute gears. With these mathematical descriptions, a general finite elements model can be created. Although this method can generate a very accurate finite elements model, this process is very difficult for complex 3D gear geometry. Since the analytical formulation focuses on gear teeth and the rim that is the part of the gear body directly under the teeth, the other parts of a gearbox may be so different that they can not be represented by a general description method.
Furthermore, writing an analysis of bending stress on the involute gear, K. Cavdar et al. (2005) developed a computer program to calculate the gear tooth bending stress. But this computer-aided calculation method is still based on the analytical formulation that is derived from DIN 3990. Although a finite elements method is presented in their research work as a comparative method, this method focuses on the theoretical gear model and is not feasible for the complex gear trains.

In order to solve the stress and deformation analysis problems of the more practical and complex gears or gear trains, some researchers began to apply the finite element method to gear analysis. By using a finite element method, Wilcox and Coleman (1995) studied the gear tooth stresses and treated the tooth as a plane stress model. This study was confined to one tooth with fixed boundaries. It was not a sufficient study since the effect upon adjacent teeth was not considered. Oda et al.(1981) further developed the studies in this domain by applying a two-dimensional (2-D) finite element method (FEM) on the analysis of the root stress distribution in a thin rim spur gear and included the effect of adjacent teeth on root stress distribution. Chong and Kubo (1985) studied the effects of the rim on the bending stress in the fillet by using 2-D triangular finite elements. With 3-D FEM, Ramamurti (1998) analyzed the gear teeth stresses of spur and bevel gears using the cyclic symmetry concept. Stress distribution along the loaded tooth, and the effects on the adjacent teeth were also studied. However, the shafts of the gear sets were not considered in their finite element models.

With the rapid development of computation capacities in the 1990's, it is possible to use a more complex finite elements method to deal with complex mechanical system and obtain more accurate solution results. Tian et al. (2003) developed a three-dimensional FE model of a spur gear system, which accommodates all the gear teeth, the gear bodies, and the two transmission shafts. The load between pinion and gear wheel is delivered by elastic frictional contact. In their work, a sub-structuring method has been used to separate the non-linear parts and linear parts. The frictional contact part of the gearbox is a non-linear problem that costs significant computing time. On the other hand, the rest of the parts of a gearbox belong to a linear problem. After the entire FE models are established, the comprehensive effects of all the gear bodies (including the transmission shafts and the supporting conditions) on the contact status are presented. Their work shows that in the analysis of complicated gear systems, results of the finite element method are consistent with the reality of an actual gear system.

From the above-mentioned, earlier studies, a finite elements analysis method has been widely studied as a robust method to analyze stress, and deformation, for the complex mechanical system, such as gear trains.

In the real-world gear train design procedure, a typical design starts with a kinematic configuration that matches the power transmission requirements and characteristics of the engine. In modern mechanical design, generally, a solid model is created by using powerful computer-aided design tools, such as CATIA, ProEngineer, or SolidWorks. To obtain reliable and direct analysis results, a finite elements model should be created by 'meshing' this preliminary design solid model. Although a geometry-based model finite element analysis makes a high demand on the computation capabilities of analysis devices and needs more time to finish the entire analysis, it can provide a complete analysis report on the whole system, without compromising or sacrificing the effects analysis of detailed features.

2.2 Optimization Method on Gear Train

The optimization design of gear trains is a reasonably difficult problem which demands the satisfaction of many design constraints. From reading the recent literature we discover that many approaches have been used for gear trains optimization.

2.2.1 Optimization on One Stage Gear Sets

The design of a standard gear train has as its objective the minimizing of the gear size for a given ratio, pinion torque, and allowable tooth strength. Savage (1982) applied computerize optimization to find the compact gear designs which balanced resistance against gear tooth bending, gear surface pitting and gear tip scoring. He developed a design methodology for sizing standard involute spur gears and applied it in order to configure an optimal design.

The developed procedure utilized standard gear geometry and optimized the design parameters to obtain the most compact standard gear set for a given application of specified speed reduction and input pinion torque. Parameters includes the gear ratio, m_g, the pinion torque, T_p, the pinion speed, Ω_p, a standard operating pressure angle of 20 or 25 deg, and standard addendum ratio and dedendum ratio. The size of gear set, which is measured by gear mesh center distance C, is another design parameter. The gear materials and their properties, such as Young's modulus, bending strength, surface endurance strength, are also important facets of the gear design problem.

Savage's method lists the design parameters available, the equality constraints which must be satisfied, and the inequality constraints which define the limits of acceptable designs. The criterion of this selection, called the merit function, is established as the center distance of the gears. The merit function is used to compare the relative merit of each possible design on an objective basis. Table 2.1 lists the primary design parameters and the relevant constraints.

Once all of these limits have been applied to the design space, it is able to survey the acceptable designs and select the optimum. The kinematic equation is used to determine the interference limit for the mesh. The bending stress and contact stress equations are used to determine the strength constraints on the design space. By plotting these curves on a graph of the pinion tooth number versus the diametral pitch, the region of acceptable designs can be established. By analyzing these designs and comparing their properties, a practical optimum design can be selected.

Table 2.1 Gear mesh parameter constraints

Parameter	Description	Equality constraint
N_1	pinion tooth number	
m_g	gear ratio	
N_2	gear tooth number	$N_2=m_g*N_1$
P_d	diametral pitch	
R_1	pinion pitch radius	$R_1=N_1/2P_d$
R_2	gear pitch radius	$R_2=N_2/2P_d$
C	center distance	$C=R_2 \pm R_1$
a_1	pinion addendum ratio	1 for standard tooth form
a_2	gear addendum ratio	1 for standard tooth form
d_1	pinion dedendum ratio	1.25 for standard tooth form
d_2	gear dedendum ratio	1.25 for standard tooth form
f	mesh face width	$f=\lambda N_1/P_d$ (λ=length to diameter ratio)
Φ	pitch line pressure angle	$\Phi=\Phi_{STD}$
E_1	pinion modulus	pinion material properties
v_1	pinion Poisson's ratio	
σ_{B1}	pinion bending design stress	
σ_{N1}	pinion surface design stress	
E_2	gear modulus	pinion material properties
v_2	gear Poisson's ratio	
σ_{B2}	gear bending design stress	
σ_{N2}	gear surface design stress	

Carroll and Johnson (1989) presented a new approach to the spur gear sets optimum problem. With some newly defined dimensionless parameters, the optimal gear set design is independent of the load and speed requirements of the gear set. The optimal design depends only on the gear ratio, m_g, and the physical properties of the materials used. By introducing a new dimensionless quantity called the Material Properties Relationship Factor, C_{MP}, the bending stress constraint can be projected onto the same dimensionless design as the two contact stress constraints. This allows the design model to be re-formed in a dimensionless design space. In this new design space, the objective function is a dimensionless "diameter". With a log-log plot of the constraints in the dimensionless space, properties for a given set of standard tooth proportions and gear ratio are shown. One of these properties is that the dimensionless diameter monotonically decreases along the contact stress constraints and the undercut constraints, and monotonically increases along the bending stress constraints for increasing number of pinion teeth. Since the location of the bending stress constraint is highly dependent on the value of C_{MP}, the location of the optimum is also dependent on C_{MP}. By using the C_{MP} value and some nonlinear equations for the number of pinion teeth, the global optimum of this dimensionless problem can be exactly determined. Then a "best" real solution can be obtained from the optimum dimensionless solution.

With the development of computers' computation speed and calculating capabilities, some researchers began to apply computerized optimal design on spur gear trains. Savage et al. (1994) continued their optimization studies on the compact spur gear sets by developing a computer-aided design procedure that applied the optimizing algorithm onto the design of the spur gear reduction. The program structure is shown in Fig. 2.1. The program includes the optimizer and two application-specific subroutines: BOUNDS and VALUES, which calculate the constraint and merit function values for each design trial. Since the optimum algorithm in this study is separated from the definition of the problem, this optimization procedure is suitable for different design problems. Another advantage of separating the analysis routines from the optimization program is the ability to modify the design conveniently at program execution and verify the characteristics of similar, more practical designs with the same optimization program.

Later in Chapter 5 and in Chapter 6, a similar methodology will be applied in the development of the genetic algorithm optimization method on gear train design.

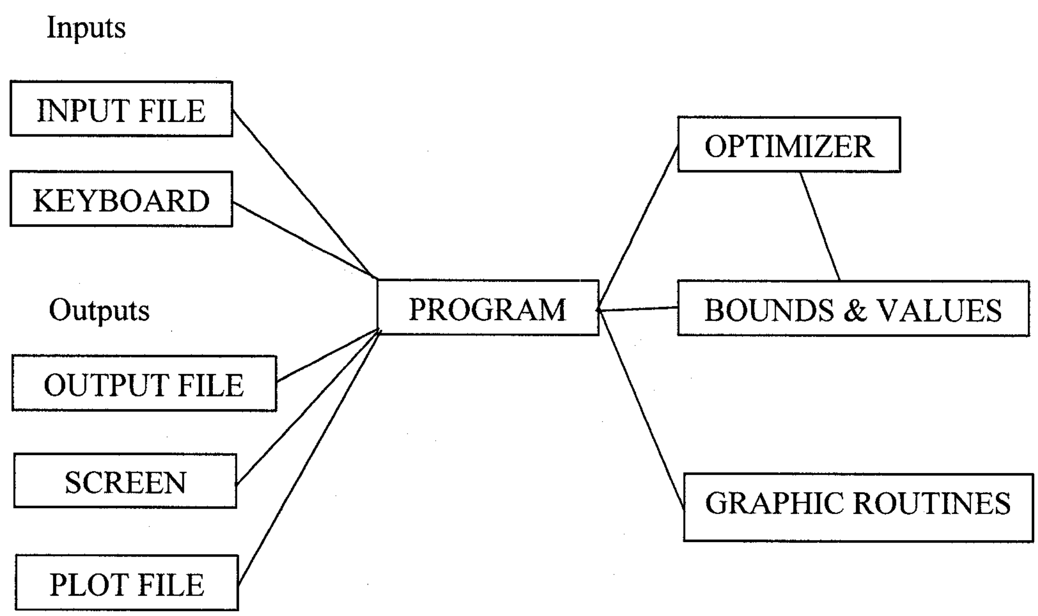


Fig. 2.1 Optimization Program Structure (Savage et al. 1994)

2.2.2 Optimal Design of Multi-stage Gear Trains

Studies made on one-stage gear sets provided us with the fundamental theories for analysis and optimization methodologies. Since, in large complex mechanical systems, multi-stage gear trains are applied in greater proportion to others, the optimal design of multi-stage or multi-speed gear sets has been a subject of considerable interest.

One of the previous studies examined a multi-parameter optimization technique used by Bush, Osman, and Sankar (1984) to solve different arrangements of a multi-speed gear train with a weighted objective function, minimizing volume and maximizing stiffness. The computer program that was developed allowed the automatically plotted speed diagram of a gear train. Based on the information supplied by the speed diagram, all possible arrangements could be formulated from one set of equations. Then the objective function was developed to minimize the sum of the distance between the shafts plus the radii of the largest gears on the input and output shafts. The dynamic properties of a gear train are closely related to the torsional stiffness. Increasing this stiffness will generally improve the dynamic properties. A weighted objective function is created by a combination of volume function and dynamic characteristic function.

Since Bush's study on the optimization of multi-stage gear train mainly focuses on the kinematic arrangement, research on this topic has been continued in order to include the gear tooth bending fatigue failure and surface fatigue failure in the consideration of optimization of gear trains. Thompson, Gupta, and Shukla (2000) include the maximum tooth bending stress σ, Eq. (2.1), and Hertz contact stress σ_H, Eq. (2.2), with the objective function of the optimization on the multi-stage gear trains.

The maximum tooth bending stress σ in the vicinity of the root fillet is estimated as:

$$\sigma = \frac{F_t P}{bJ} K_v K_o K_m \tag{2.1}$$

where F_t represents the tangential gear force

J Lewis spur gear geometry factor

P diametral pitch

b gear face width

K_v velocity factor

K_o overload factor

K_m mounting factor

The Hertz contact stress σ_H at the tooth surface is estimated as:

$$\sigma_H = C_p \sqrt{\frac{F_t}{bdI} K_v K_o K_m} \tag{2.2}$$

where C_p represents the elastic coefficient of the material

d diameter of pinion or gear

I geometry factor

b gear face width

K_v, K_o, K_m as the same definition as in the equation Eq. (2.1)

In addition to the constraints on bending fatigue and surface fatigue, a number of geometry constraints are also involved. These constraints include range of face width *b* and maximum allowable addendum diameter. The presence of conflicting objectives and/or constraints is apparent. The problem of simultaneous satisfaction for these constraints is a feasibility, or constraint, satisfaction problem. The optimization problem of gear sets becomes a multi-objective optimization by studying the trade-off between fatigue life and minimum volume. The solution for such a problem is not a single optimum design, but rather a collection of optimal designs, each of which is optimal for a given surface fatigue lifetime. (The topic of trade-off method will be discussed in section 2.3.)

2.2.3 Genetic Algorithm on the Optimization of Gear Trains

Genetic Algorithms (GA) are adaptive heuristic search algorithms premised on the evolutionary natural selection theory of Charles Darwin. The principle of Genetic Algorithms is to simulate the processes within a natural system which are necessary for evolution, which follow the principles of the "survival of the fittest". As such they represent the intelligent exploitation of a random search within a defined search space to solve a problem.

First pioneered by John Holland in the 1960s, Genetic Algorithms have been widely studied, experimented with, and applied, within many fields in the engineering world. Not only do GAs provide alternative methods to solve optimization problems, they consistently out-performs other traditional methods when trying to solve most of the problems presented. Many of the real-world problems have involved finding optimal parameters, which might have proven difficult for traditional methods but are ideal for GA.

The Genetic Algorithm method starts with a population of individuals chosen randomly and coded as chromosomes by using binary numbers. Then the algorithm generates new populations from the former individuals which are able to be more adaptive to the well-defined problem environment. The best performing individuals who demonstrate a better fitness have a great possibility of being chosen to become the parents of the new generation. The creation of the offspring is carried out by genetic operations, such as crossover and/or mutation. With the process continuing, the better individuals are kept and the poorer individuals are eliminated, which propels the better individuals forward to obtain the best fitness. After the whole process finished, the "best" candidate is treated as the optimization result for a given problem.

Applying Genetic Algorithms onto the optimization of gear systems has been a considerable interest in the past. Yokota, Taguchi and Gen (1998) proposed a solution method for optimal weight design of gears using a genetic algorithm. They formulated an optimal weight design problem of a gear and defined it as a nonlinear integer programming problem concerned with the constrained bending strength of the gear, tortional strength of the shafts and each gear dimension. Each design variable of a spur gear is represented by the gene of chromosome which is an integer-valued decision variable. The initial population is randomly generated by satisfying the earlier defined constraints. With the execution of crossover and mutation on the chromosomes, new populations are generated and their fitness is evaluated. The best chromosomes are chosen for the next generation and the procedure is repeated until the best optimization result is found.

Another application of a Genetic Algorithm method implemented on the optimal design of gears is presented by Saravanan et al. (2001). In this study, they defined a suitable, combined objective function to minimize both error and weight, and thereby maximize efficiency. This objective function is a multi-objective function, which we will discuss later in the section 2.3. The Genetic Algorithm is applied onto the design example of a gear-set for an IC engine to transmit the power. From the results of this example, the results obtained by GA are much better than the values obtained by the conventional method. Both error and weight were reduced to a greater extent than those results achieved by the conventional method. As a conclusion, the gear design optimization problem can be solved efficiently by the Genetic Algorithm method.

There are still many other applications of Genetic Algorithms on the optimization of gear trains. All of these studies show the powerful capabilities of GA to determine the optimal combination results of design variables for a given problem. It has been proven that when a complex optimization problem can not be solved easily by conventional method, GA is a reliable and convenient method for such optimization problems.

As we can see in the earlier studies, most of the optimization methods of Genetic Algorithms for gears or gear set designs are based on the mathematical analysis of gear volume or stress constraints. As mathematical analysis comes from the standard gear set design and it is very difficult to write an accurate description for a complex gear system, these methods are not applicable for a real-world gear system. A finite element based analysis incorporated with an optimization Genetic Algorithm will be presented in this thesis. With the rapid development of computer technology, the FEA method, which usually cost a great deal of computation time in the past, becomes acceptable in regards to time-related effectiveness.

2.3 Development of Multi-objective Optimization

The optimization of a gearbox has been always a very complex procedure. The design of a gearbox involves optimization on gear geometries, kinematic behaviors, strength analysis, and if necessary, dynamic studies. The geometries optimization also could include integer valued variables, discrete valued variables, or continuous real-valued variables. The real-world problem becomes further complicated due to the presence of the conflicting optimization objectives.

Earlier researchers tried different methods in order to tackle such complex problems. A multi-step optimization procedure is usually adopted. In this method, the overall problem is broken down into tractable sub-problems (1990). In each step, a part of the design problem is solved by fixing other variables to the reasonable values. Then a further part of the design problem is solved by fixing the previous design variables to the values obtained from the first step. This procedure continues until the whole problem is solved. Although this method can solve a more complex problem, with more than one objective, the optimal results may not provide the real, optimal design for such problem.

To solve the minimum volume design problem of multi-stage spur gear sets, Thompson, Gupta and Shukla (2000) demonstrated a trade-off analysis between the surface fatigue life, and minimum volume, using a basic multi-objective optimization procedure. The multi-objectives of a problem is posed as: $f_1(x), f_2(x), \ldots, f_r(x)$. Then the multi-objective optimization problem is posed as follows

$$\min \{ f_1(x), f_2(x), \ldots, f_r(x)\} \tag{2.3}$$

By incorporating the weighting factors $\alpha_1, \alpha_2, \ldots, \alpha_r$ for each objective, the problem is shown as

$$\min \{\alpha_1 f_1(x) + \alpha_2 f_2(x) + \ldots + \alpha_r f_r(x)\} \tag{2.4}$$

The answer obtained from this multi-objective optimization analysis is not a single optimum design but rather a 'trade-off' Pareto set which is a collection of optimal designs. With this information the designer can judge overall optimization trends and assess the penalty of one objective in regards to the other objectives.

To optimize several objectives simultaneously on spur gear sets, Wang, H. and Wang H. P. (1994) proposed a multiple-objective optimization methodology for spur gear design. The objectives in this research are as follows:

- minimum size of the meshing gear set;
- minimum weight of the meshing gear set;
- minimum tooth deflection; and

- maximum secure life of the gear set.

The size of gear sets can be calculated by the central distance, and weight can be obtained from the geometries volume of gear sets. Tooth deflection and useful life can be calculated from corresponding formulas as well. The constraints such as bending strength and surface stress can be written in AGMA standards. By employing a solution procedure that is called the Modified Iterative Weighted Tchebycheff (MIWT) method, this kind of optimization problem can be solved. The advantage of this procedure is that it operates in a fixed number of iterations, and, for each iteration, a fixed number of solutions is generated.

Since classical multi-objective optimization methods require the designer to supply a weighting factor or a preference factor for each objective, the outcome of this scalarized multi-objective optimization process is usually a single optimal solution. In order to obtain a set of Pareto-optimal solutions, different weighting factors for the objectives have to be applied in many cases. To further studies in the multi-objective optimization, Deb and Jain (2003) presented a multi-objective evolutionary algorithm (MOEA), which is capable of solving the original multi-objective problem as well as finding multiple 'non-dominated' solutions in a single simulation run. The proposed technique – the modified non-dominated sorting Genetic Algorithm (NSGA-II) can handle the complexities which might arise in an optimization problem. NSGA-II begins its search with a population P of N randomly, creating solutions. The population members are divided into a set of subpopulations. Thereafter, each solution in a subpopulation is assigned a crowding distance value, equal to the sum of the difference between the two neighboring solutions in each objective. The crowding distance value is small if the neighboring solutions are close to a solution in the objective space. After three operations – tournament selection, recombination and mutation – are applied on the population, the offspring population is generated, which is combined with the parent to build a new population of size $2N$. The new population is sorted and selected according to the non-domination level. The final accepted population becomes the new parent population and this procedure is repeated till a termination criterion is satisfied. The results converge closer to the Pareto-optimal solutions with a well-distributed set of solutions. Finally, the obtained solutions are analyzed to discover important design principles related to an optimal design of a gearbox.

2.4 Integration of Analysis and Optimization Tools

Mechanical systems are very complex, and design and analysis processes usually can not be finished by using a single tool. In modern design, different kinds of implements, such as personally-developed solving tools or commercial software, are widely used to facilitate various phases of design, analysis and optimization. Since these tasks are performed on the different stages, an integration method should be constructed to facilitate communication between these phases, and then, to establish a batch mode and realize the complete design procedures automatically. Although some commercial programs have combined design, analysis and optimization in a single system, there are still many limitations in regards to their applications. To deal with this practical complex problem, engineers have had to repeatedly manipulate design models found in the different CAD, CAE and optimization software.

The integration of finite element analysis, and optimum design on the gear system attracted much attention in earlier research. Li et al. (2002) established a batch module which consists of I-DEAS, ABAQUS and MOST to do the pre-processing, the numerical-solving and the optimization, respectively. The module called "integration of finite element analysis and optimum design" is able to search for contact nodes and elements and automatically define the contact surfaces for contact analysis. Fig. 2.2 shows the contents of the integrated module with the tasks of all components. In this integrated system, I-DEAS was adopted because of its capability to create a complex geometrical model. Although I-DEAS has the FE solver and optimizer, it does not support sliding contact analysis. ABAQUS is a general finite element software with a

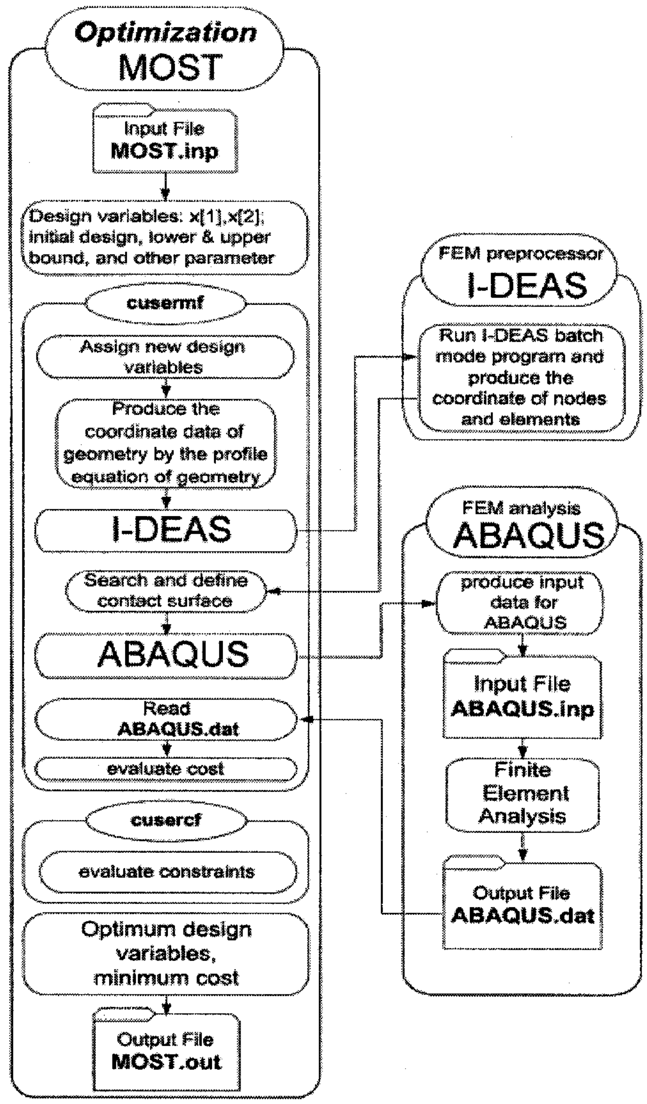


Fig. 2.2 Components of the integrated module and its interface

powerful capability for processing sliding contact problems. It can solve sliding contact problems without point to point contact constraint. MOST is a C-based optimizer that can solve both continuous and discrete design variables optimization problems. Another advantage is its ability to deal with multi-objective optimization.

As presented in this paper, the integrated system can automatically construct the geometrical model, analyze contact stress, and solve the optimal solutions when gearing parameters are input. Tested on two examples, a simple gear-pair system (one pinion and one gear) and a planetary gear system, this module can generate good results. It must be mentioned, however, that, although MOST has three modules to solve problems that contain continuous variables, non-continuous variables, and are multi-objective, this integration system only composes in the continuous variables module and can only solve single objective optimization, which makes it impractical for solving more complex gear system optimization problems.

To overcome the problems in design and optimization of worm gears, Su and Qin (2003) developed another comprehensive computer-aided approach, which integrated multiple techniques including numerical analysis, three-dimensional simulation, and finite element analysis. Fig. 2.3 shows that this approach included three modules: numerical analysis, three-dimensional simulation (Pro/Engineer) and finite element analysis (ANSYS). The numerical analysis module calculates the tooth geometries which base on the differential geometry theory. In the second module (three-dimensional simulation), 3-D models are generated by CAD package ProEngineer, either from the numerical analysis results in the first step, or by simulating the manufacturing process. Thereafter, the finite element analysis module calculates the mesh stiffness, tooth bending stress and surface contact stress. Since the geometry model is created from the previous numerical analysis, this process is complicated and can not be parameterized. Importing the IGES format of the geometry model from ProEngineer to ANSYS requires additional manual processing, such as creating the volume, and defining the element types and materials, which means it is not really an automatic design system. Furthermore, this integration approach does not include the optimization module, which is an important part of the design procedure for mechanical systems in the aeronautic industry.

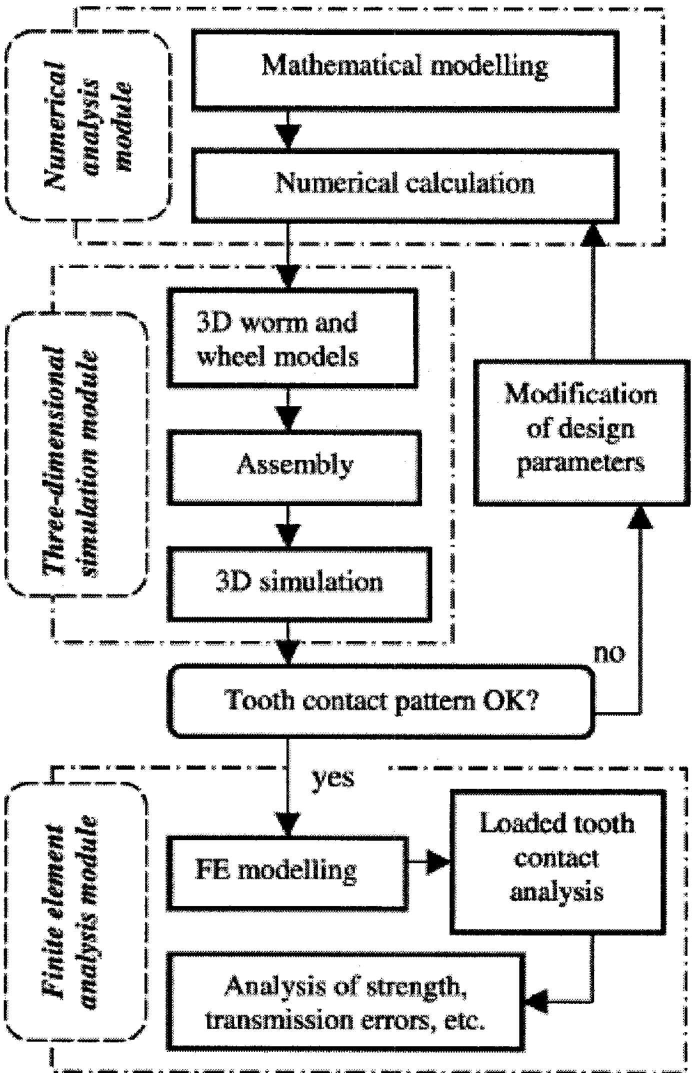


Fig. 2.3 Flowchart of the integrated approach

Although the above-mentioned integration explorations all demonstrate various limitations in the automatic analysis, and optimization design for real and complex gear systems, they provide us with many good ideas and valuable experience which will assist us as we seek a more practical approach for integrating different design, analysis and optimization tools.

2.5 Conclusion

This chapter has reviewed the historical studies of the different aspects of gear system research. The design of a gear system is a complex procedure that involves many disciplines. Each aspect, such as gear analysis or design optimization, can be extended into a wider research field.

The first section introduces the different analysis method of gear sets. Gear analysis includes geometry analysis, kinematic analysis, stresses and lifetime analysis, and dynamic analysis. This section focuses on the gear stresses and deformation analysis methods, as well as the numerical analytical method and the finite element method. For the single gear or a simple gear set, the numerical method, which is based on formulas come from mechanical theory, is accurate and easy to accomplish. But for a complex gear system, such as epicyclic gear sets or multi-stage gearboxes, the finite element method provides the most powerful and complete system solution. This is why the finite element method has become an important gear analysis tool in the recent years.

The second part gives a global view of the popular optimization methods for the design of gear systems. These methods have been developed while working with the optimization of the single stage gearbox, the optimization of the multi-stage gearbox, the conventional mathematical optimization which relies upon plotting figures, as well as the stochastic-based Genetic Algorithm optimization method. Since the Genetic Algorithm is suitable for optimization with multi-design variables combined with continuous and non-continuous variables; and has capabilities for searching an entire design space, it has also been adopted in this study as the main optimization method used for the optimal design of gear sets.

The third section discusses earlier research of the multi-objective optimization methods for gear train design. Multi-objective optimization study is actually a trade-off of different optimization objectives. It provides a set of optimal design solutions rather than a single optimal solution. This topic is still a complicated optimization study that has attracted considerable attention even in the recent years.

The last part presents the integration of different function tools during gear design procedures. It establishes the intention to automatize or partly automatize the gear design, analysis and optimization procedure.

From the earlier studies we can conclude that the finite element method is still a powerful and complete gear analysis method; the Genetic Algorithm can provide a comprehensive exploration of gear system optimization, and, the integration of analysis and optimization tools remains a task under scrutiny. The research work in this thesis will surround these main points to provide a feasible procedure of finite element analysis on the epicyclic gear train, a Genetic Algorithm optimization module fitting the finite element analysis method, and an integration solution to incorporate the FEA module and the GA optimization module.

CHAPTER 3

FINITE ELEMENT ANALYSIS ON EPICYCLIC GEAR TRAINS

This chapter presents a general finite element analysis (FEA) procedure on epicyclic gear trains. Due to the compact size of the part's geometry and the large-range gear-transmission ratio, the epicyclic gear train is one of the most important power transmission systems and is widely used in the aircraft and automobile industries. Devices mounted in the aircrafts must prove to be of high strength and stiffness and, as well, to be as light as possible. Unlike other civil products, which can be safely designed by using safety factors based on experience or standards, the mechanical systems in aircrafts need a complete certification by analysis as well as a bench test in order to acquire the optimal design. Traditional analysis methods based on analytical formulation give us a theoretical basis and reflect a great deal of experience, but normally do not take into account many details featured in the products. These details, such as a functional hole, or manufacturing features, may generate a certain amount of difference in the analysis results when comparing the analytical method with experimental results. This difference is critical in aircraft products and may invalidate the product if the safety factors are considered low. Thus, a finite element analysis on the final design including all its features is an ideal analysis method due to its capability to provide complete systematic analysis results.

3.1 Global View of an Epicyclic Gear Train

A simple epicyclic gear train is a planetary gear arrangement and consists of one or more planet (epicyclic) gears meshed and rotating around a central sun gear. The planet gears are also meshed and rotate within an internal ring gear. A planet carrier is connected to the planet gears and is designed to rotate on the same axis as the sun gear. (Fig. 3.1)

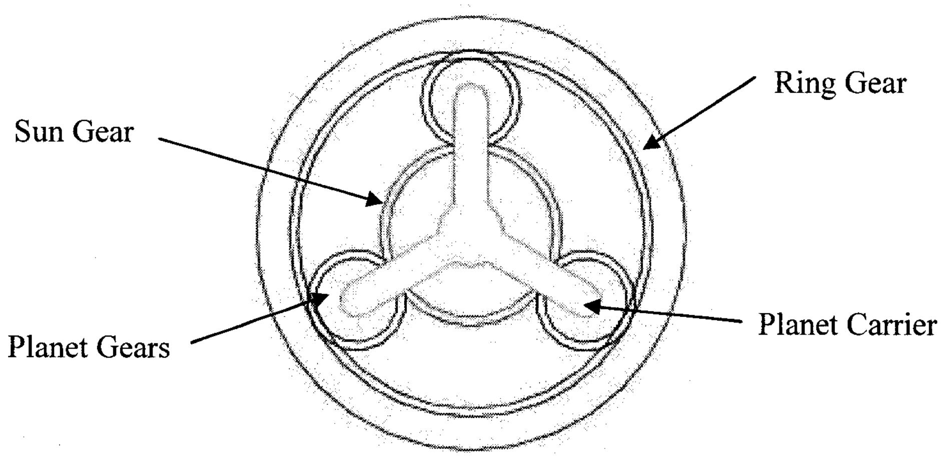


Fig. 3.1 Typical Epicyclic Gear Train

With different members locked, an epicyclic gearbox has different operation modes. The most classic arrangement is with the ring gear fixed, the sun gear designated as the power input end and the planet carrier as the power output end.

3.2 Current Analysis Method

The epicyclic gear train is a complex mechanical system in a very compact solution. A complete epicyclic gear train can have more than one hundred parts mounted in a reduced space. The interaction among these parts makes the stress analysis very complex. Generally, a finite element analysis method is applied using powerful commercial analysis software. Currently a typical analysis procedure is:

- Preliminary design based on the power transmission ratio, speed reduction ratio and other kinematic requirements;
- Build the 3D geometry models of each part in CAD system from the preliminary design;
- Assemble all the parts of the epicyclic gear train with their relationship in CAD system;

- Mesh the 3D assembly model for finite element analysis within the CAD system or in a specialized mesher;
- Import the meshed assembly model into analysis tool;
- Define contacts, loads and boundary conditions;
- Launch a simulation, obtain the results;
- Post-process to locate the maximum stress and maximum deformation on the critical points;
- Repeat above steps if the stress or deformation criteria are not fulfilled.

This procedure can satisfy any general design. But for advanced optimal design that requires global weight control for materials that weight the least possible amount, another optimization tool has to be incorporated. This makes the whole design procedure more complex and communication problems may arise. For this project, we chose ANSYS Workbench as the FEA software containing modules for geometry model design, finite element analysis and powerful optimization. It is also a comprehensive and multi-disciplinary analysis tool that can deal with static mechanics, as well as heat and fluid problems. Like any other commercial software, its powerful and multifunctional capabilities do not mean that it can analyze or optimize the epicyclic gear train automatically. One of this researcher's tasks is to create a general procedure for the stress and stiffness analysis, as well as the optimization of the epicyclic gear train in the environment of ANSYS Workbench.

3.3 Simplification of the Real Gear Train

The epicyclic gear train is a complex mechanical system that has many sliding contacts. Sliding contacts lead to non-linear analysis and increase the computing time manifoldly. Furthermore, a non-precise definition of contacts will result in non-convergence within the FEA and the solution may never be obtained. The reason for these problems can not be found easily. Thus, to avoid above situation and in order to propose a robust procedure of analysis, simplification of gear train is required in the preliminary step of the procedure development.

3.3.1 Reduce Non-crucial Parts and Features on the Gear Train

An example of a complete epicyclic gear train is shown in Fig.3.2. It has more than one hundred sixty parts when assembled together and has full detail of design features and manufacturing features.

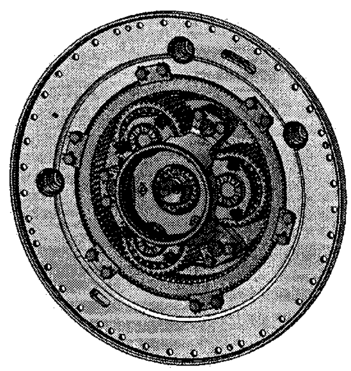


Fig. 3.2 A complete epicyclic gear train

Directly applying the FEA to this assembly is not being an easy task and yet the IT resources implied would be very important to do so. The computation time is considerably long and the results are not predictable. In the first step of simplification, all the non-crucial parts are eliminated, such as gear train housing, all the bolts and washers, and all the bearings. Only the most important parts are kept, such as the sun gear, the planet gears, the planet carrier and the ring gear. All the bolt-connections will be replaced by bonded contacts. All the bearing supports will be replaced by cylindrical supports. By this simplification, the number of components of this assembly has been decreased to less than twenty.

3.3.2 Re-creation of CATIA Model for Finite Element Analysis

Simplification as processed in the previous section is not enough. Running a finite element analysis solution for such an assembly, using an ordinary desktop still costs hours of computing time. Another limitation we face is the fact that the total number of nodes of the finite elements model can not exceed 128,000 in the academic version of ANSYS Workbench which we used. Since our purpose is to develop a FEA procedure for epicyclic gear trains, the detailed features on the parts are eliminated. Once the procedure has been proven to be able to execute a full FEA solution; adding these detailed features just extends the computing time, but does not cause any major difficulty.

To keep the main geometry features of epicyclic gear train, a simplified geometry model is created in CATIA V5 environment (Fig. 3.3). This geometry model has the complete involute tooth profile, which can simulate the real kinematics between the meshed teeth. Other features like chamfers, un-functional holes and slots are not shown in this model. After this second simplification, the whole gear train assembly has only six parts: which are one sun gear, one ring gear, three planet gears and one planet carrier. A simplified gear train geometry model, ready for a finite element analysis, is shown in Figure 3.3.

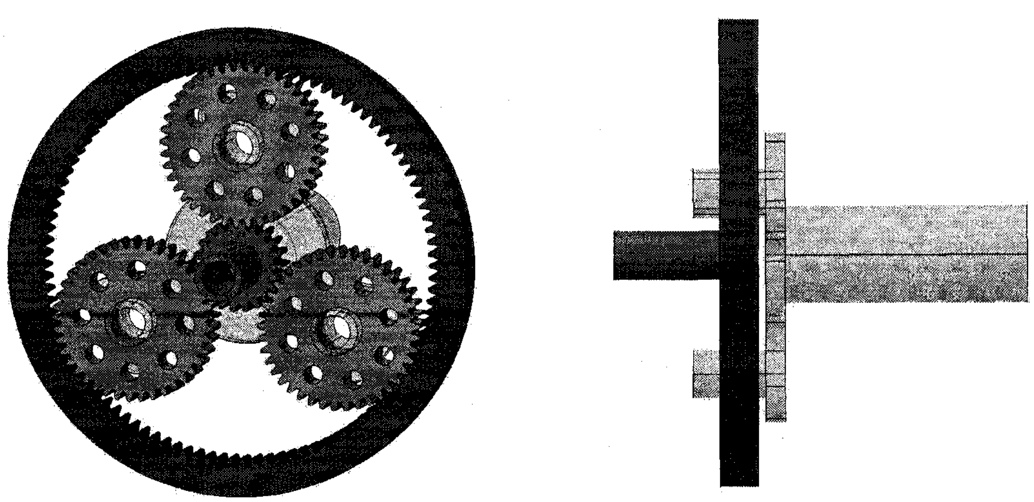


Fig. 3.3 Simplified epicyclic gear train model

3.4 Finite Element Analysis on Simplified Gear Train

This section presents an application of finite element analysis on the simplified gear train geometry model. The gear train model is initially built in CATIA V5 and then imported into ANSYS Workbench. Within ANSYS Workbench several operations are performed:

- Reduce the global geometry model to one third;
- Realize the meshing and apply the boundary conditions;
- Solve the simulation and post-process the results.

3.4.1 Import CATIA Model into ANSYS Workbench

Once a simplified geometry model is ready in CATIA, it can be imported into ANSYS Workbench with the help of an associated reader. In a previous study of Su and Qin (2003), although a similar CAD tool – Pro/Engineer and ANSYS are used to achieve the CAD modeling and FEA solution respectively; the IGES model format transferred from one to another needs some additional processing. These manually fulfilled processes, such as re-building the volume, defining the element and materials type, make the whole procedure more complex and subject to error. The associated reader between CATIA V5 and ANSYS Workbench can keep all the geometry information, material information and the relative position of each part of the system. Another advantage is that it can transfer the defined parameters from CATIA into ANSYS Workbench. This is a very important feature for the next step in optimization, because optimization needs the manipulation of geometric parameters transferred from the geometry model. For this reason, the advantage of using simplified geometry model is obvious. Importing a simplified geometry model costs a few minutes' time whereas importing a full featured geometry model needs more than thirty minutes.

3.4.2 Further Simplify the Gear Train Model in ANSYS Workbench

After importing the geometry model into ANSYS Workbench (Design Modeler module), another important step is to further simplify the model. Since the epicyclic gear train is a cyclical symmetry system, a symmetry section with the appropriate symmetry boundary condition will lead to the same FEA result, but will dramatically decrease the number of finite elements and consequently save the computing time. Having considered the three planet gears in this epicyclic gear train, a 120 degree of symmetry section is cut out from the whole system. Secondly, we retain only one or two teeth on a gear that are to be meshed with the corresponding teeth on the other gears, and the adjacent teeth of the meshed teeth that have effects on the stress analysis; all other teeth can be removed from the geometry model for the approximate FEA solution. With these two steps of simplification, the gear train model presents the most reduced computing time for the FEA solution. (Fig. 3.4)

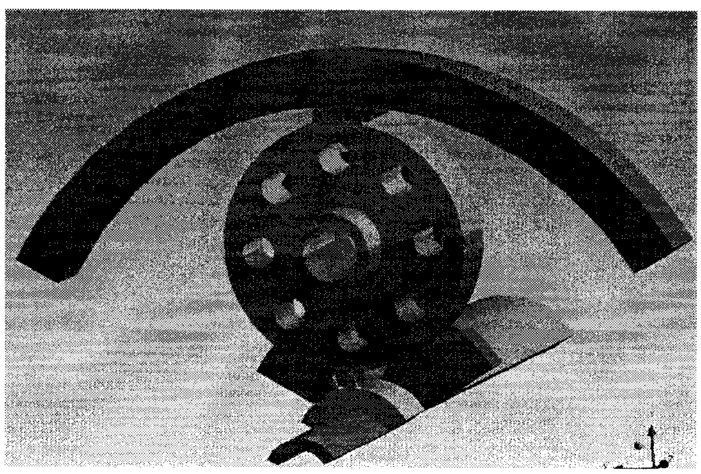


Fig. 3.4 Simplification of gear train model in ANSYS Workbench

3.4.3 Volumes partitions for hexahedral mesh

Before the simplified geometry model is transferred from Design Modeler module to simulation module of ANSYS Workbench for the FEA solution, a process called Cut-

Glue (CG) will be applied onto the gear teeth section. CG cuts out the gear teeth region from the gear, then it glues the gear teeth and gear body together into one part, as in its previous form. The part is still treated by ANSYS as one part and continuity of mesh is assured (see Fig. 3.5). If this CG process is not applied on the gear model, meshing on the original gear can only generate free tetrahedral elements, which are not accurate in the FEA solution. Furthermore, the number of elements becomes large so the computing time increases. But with this CG process on the gear teeth, due to a virtual boundary between gear teeth and gear body, a mapped hexahedral can be easily obtained and good control of the tooth mesh is achieved.

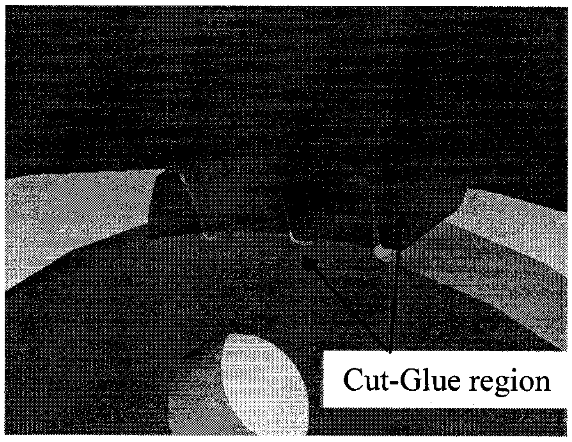


Fig. 3.5 Cut-Glue process on the gear teeth section

3.4.4 Mesh Gear Train Model and Apply Boundary Conditions

Mesh definition:

Once the gear train model has been simplified and processed in Design Modeler module, it can be imported into Simulation module for the FE analysis. This section shows the meshing and environment definition of the geometry model before running a FEA solution. Since the geometry has been properly partitioned, the procedure of meshing is quite simple. Below are the main points of meshing:

- Set a global element size for the whole gear train except for the gear teeth sections;
- Set the local refined element size for the gear teeth sections for accurate FEA results;
- Define identical meshing on the opposite faces of the cyclical cut by using Match Face Mesh command (see Fig. 3.6). Install cyclical symmetry that is necessary to obtain the proper behavior when a torque loading is present.
- Apply the meshing

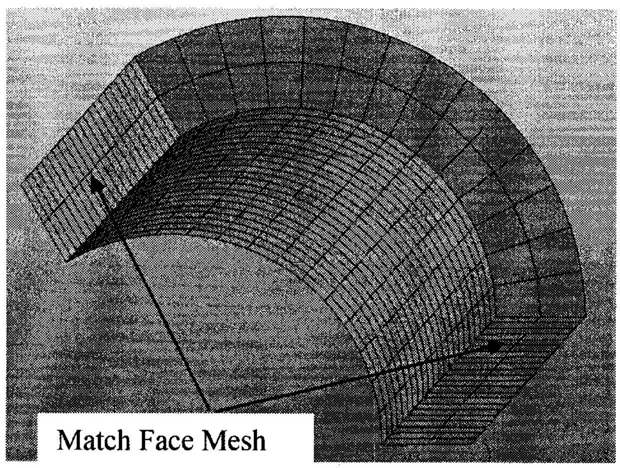


Fig. 3.6 Match Face Mesh on opposite cut faces for cyclic symmetry

Contact definition:

The next step is to define the contact pairs. There are three contact sections on an epicyclic gear train: contact between sun gear teeth and planet gear teeth, contact between planet gear teeth and ring gear teeth, contact between the planet gear central hole and the planet carrier pin. To simulate real kinematic conditions, these three contacts are defined as frictionless sliding contacts. It has to be mentioned that the sliding contact between the planet gear central hole and planet carrier pin is not defined by normal GUI (Graphical User Interface) method, but with an APDL (ANSYS Parametric Design Language) command. Since the meshing patterns are not similar on the interface of the two parts, the contact defined by GUI is not well established to simulate the sliding status. With the APDL command – CPINTF, the degree of freedom (DOF) of the coupled nodes on the interface is defined as tangentially free. The effect of this definition is the same as the frictionless sliding contact defined in GUI. (See Annex for the block of commands containing the CPINTF command.)

Loads and boundary conditions:

Once the meshing is ready and contacts are defined, the next step is to apply the loads and boundary conditions (Fig.3.7). There are several tasks in this step:

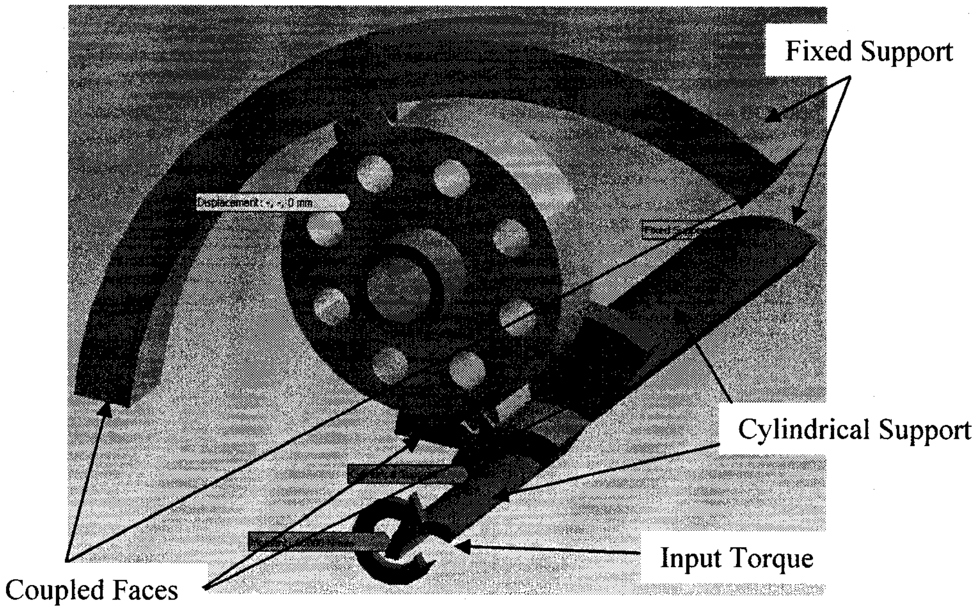


Fig. 3.7 Boundary Conditions on Epicyclic Gear Train

- Apply a torque (moment) on the end face of sun gear shaft;
- Apply fixed support on the outer face of ring gear and the end face of planet carrier shaft;
- Apply cylindrical support on the surface of sun gear shaft and the surface of planet carrier shaft;
- Couple the two side faces of the cyclically symmetric model on the whole system, with APDL command – CPCYC.

Modeling issues:

As a result of the experiment a conflict is found when applying the CPCYC command. The conflict comes from the two edges on the end face of the sun gear shaft. These two edges are possessed in common by the CPCYC command and by torque definition. This conflict is identified by the Newton-Raphson residuals in a non-converged simulation. To solve this problem, selected nodes on the intersection between the two faces (the two common edges) have to be removed from the total selected nodes. (Fig. 3.8) (See Annex for CPCYC command).

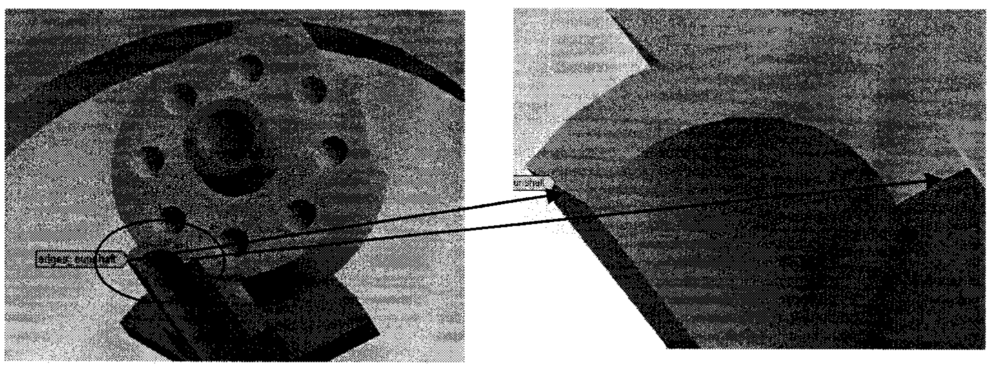


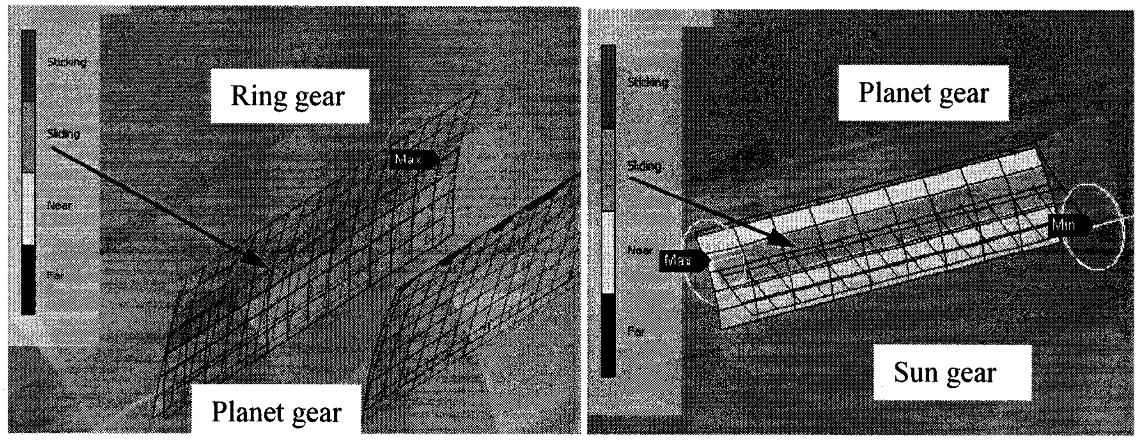
Fig. 3.8 Conflict on the two edges addressed by two commands

3.4.5 Solution and Results Analysis

After several actions on tuning parameters of the solver (memory, total steps number, initial steps number, type of solver, etc), the non-linear analysis solution is obtained in approximately seven minutes on the Intel Pentium D platform with dual-core 3.0GHz CPU. The result analysis composes gear teeth contact status, teeth surface contact stress, teeth bending stress and planet carrier pin stiffness. These are the main results which are commonly read for this type of analysis.

- Gear Teeth Contact Status and Penetration

Gear teeth contact status can be verified by Contact Tool in the Simulation module. As shown in the result of contact status (Fig. 3.9), "sliding" proves a correct simulation of frictionless contact between the meshed teeth. It also demonstrates that although two pairs of teeth contact are defined in ring-planet gear contact zone, there is only one pair of engaged teeth at one time.



a. Teeth contact between ring - planet gear

b. Teeth contact between sun - planet gear

Fig. 3.9 Contact Status of Engaged Teeth

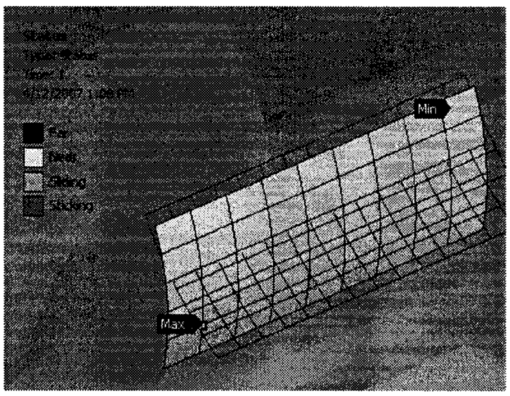
The contact formulation method used is called Augmented Lagrange. In the real world, the contacting bodies do not inter-penetrate; therefore the contact relationship between two surfaces should be established to prevent them from passing through each other in the finite element analysis. ANSYS offers several different algorithms to prevent interpenetration by using a contact "spring" at the contact interface. The spring stiffness is commonly called the *contact stiffness*. The spring will generate a contact force F at the interface for equilibrium.

$$F = k\Delta$$

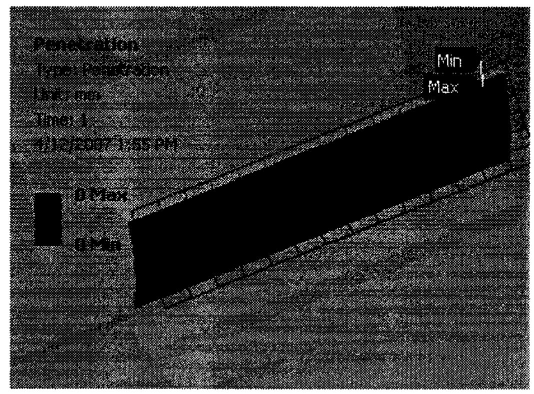
where k is the contact stiffness;

Δ is the amount of penetration.

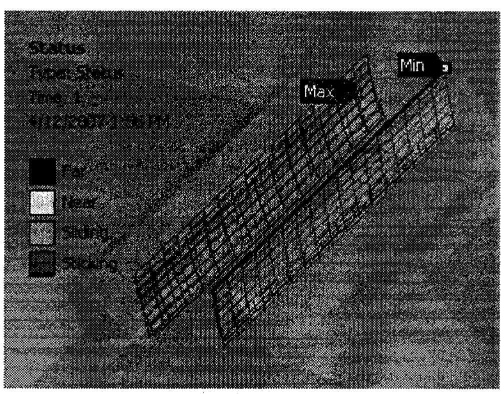
Since the physical, real world, contacting bodies do not inter-penetrate, in order to achieve accuracy, the goal is set to minimize the amount of penetration that occurs at the contact interface. Minimum penetration gives maximum accuracy, while too high a measure of contact stiffness leads to convergence difficulties. Minimum penetration, according to the Augmented Lagrange algorithm which we chose to work with, is allowed in order to stabilize the solution and obtaine a converged solution. If the contact stiffness is too great, a slight penetration will generate an excessive contact force, potentially throwing the contacting surfaces apart in the next iteration. Fig. 3.10 presents the contact status and penetration value in both sun-planet and ring-planet contact zone with the Pure Lagrange method (nearly zero penetration).



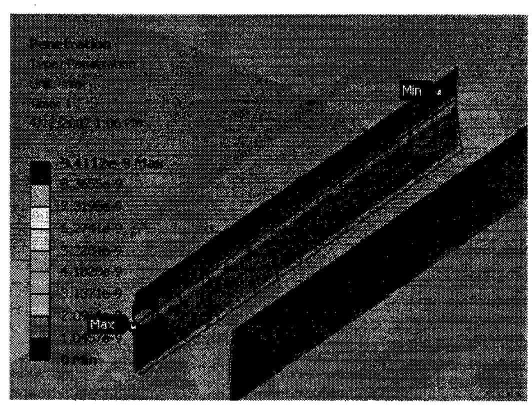
a. Contact status in sun - planet gear zone



b. Penetration in sun - planet gear zone



c. Contact status in ring - planet gear zone



d. Penetration in ring - planet gear zone

Fig. 3.10 Contact status and penetration with Pure Lagrange

Thus, stiffness which is too great leads to oscillation convergence, and often to outright divergence (Fig.3.10 a and b). In the Augmented Lagrange method, contact pressure is increased during equilibrium iterations so that final penetration is smaller than the allowable tolerance. This method allows greater penetration than the Pure Lagrange method (nearly zero penetration) and less than the Pure Penalty method (which has a large degree of penetration). It provides better contact conditions than Pure Penalty and can prevent the convergence difficulties which are found in Pure Lagrange method; but normally it requires more iterations than the Pure Penalty method. A comparison of the three contact algorithms is shown in Table 3.1. A conclusion can be drawn that the Augmented Lagrange method prevents the convergence problem by increasing the penetration, but can not guarantee that it will result in a smaller penetration than that which is accomplished by the Pure Penalty algorithm.

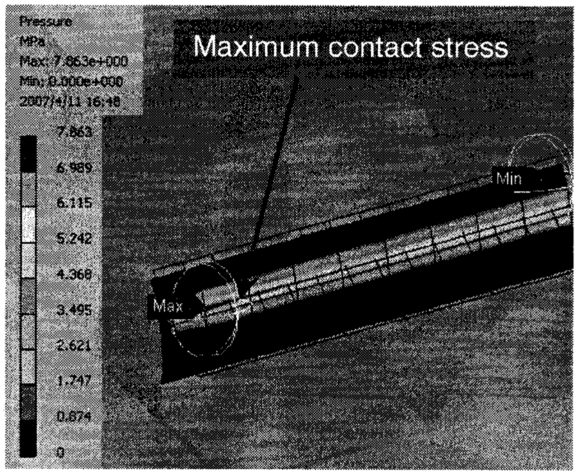
Table 3.1 Comparison of different contact algorithms

	Pure Lagrange		Augmented Lagrange		Pure Penalty	
Contact zone	Status	Penetration (mm)	Status	Penetration (mm)	Status	Penetration (mm)
Sun-planet	Near	0	Sliding	1.029e-4	Sliding	1.029e-4
Ring-planet	Sliding	9.411e-9	Sliding	5.733e-5	Sliding	5.733e-5

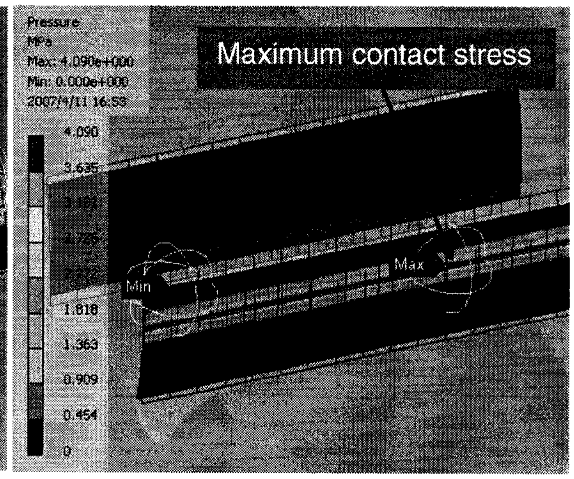
- Surface contact stress and teeth bending stress

Teeth surface contact stress and teeth bending stress are the two most important indicators in the evaluation of a gear train lifespan. Most of the gear system failures occur due to teeth surface pitting or teeth root cracking. From the result of the FEA solution, maximum contact stress and maximum bending stress can be obtained conveniently according to the gear theory. (Fig. 3.11)

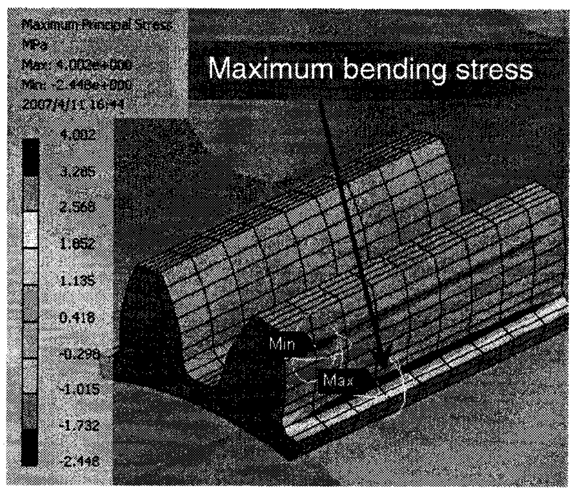
Fig. 3.11 shows that maximum contact stress occurs on the section of the contact line and that maximum bending stress occurs on the tooth root. Although contacts are defined as surface-to-surface contacts, the contacts actually occur as line-to-line contacts. The results of FEA respect the contact and bending stress occurrences in actual gear teeth contact situations.



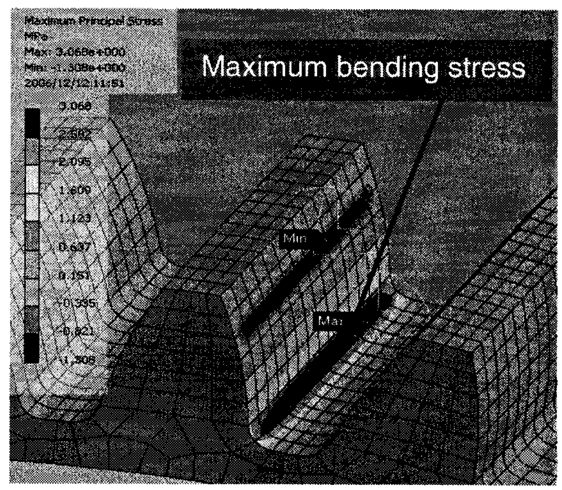
a. contact stress in sun - planet gear zone



b. contact stress in ring - planet gear zone



c. bending stress in sun - planet gear zone



d. bending stress in ring - planet gear zone

Fig. 3.11 Maximum contact stress and maximum bending stress

A good distribution of contact pressure in ring – planet gear zone is shown in Fig. 3.11(b). The contact pressure is uniformly distributed along the contact line and maximum contact pressure is located in the middle of the contact line. Since the carrier pin stiffness can largely affect the distribution of contact pressure, the results of Fig. 3.11(b) prove that the carrier pin stiffness is correct for this zone.

Meanwhile, the maximum contact pressure in the sun – planet gear zone is located on one end of the tooth face (see Fig. 3.11(a)). This is explained by the fact that the gear shaft is stiffer at its input end (due to higher metal content) than at the cantilever end where the gear is located. Further investigation of the dispersion of the contact stress along the tooth face should indicate if the design needs to be revised. If high local contact stress is revealed, or if a limited contact patch is shown, we might observe rapid damage to the surface teeth. Generally, a proper shaft design or applying a crowning to the gear teeth will avoid such phenomenon.

- Contact pressure convergence study

A convergence study on the contact pressure is performed at the sun – planet gear contact zone. The global element size is set as 16 mm. The local element size at the sun – planet gear contact zone is reduced from 4 mm to 2 mm. Since the element size greatly affects the results of the finite element analysis, a convergence study is important to find real contact pressure in the simulation model. The results of the convergence study on sun – planet contact pressure are shown in Table 3.2.

Table 3.2 Contact pressure convergence study (sun-planet contact zone)

Model	1	2	3
Element size (contact zone)	4 mm	3 mm	2 mm
Contact pressure	56.06 MPa	76.45 MPa	73.39 MPa
Bending stress	34.83 MPa	34.63 MPa	36.99 MPa

Table 3.2 shows that the contact pressure converged during the reducing of element size on the sun – planet contact zone, whereas the bending stress was stable when the element size was changed. The contact pressure is more sensitive to the element size than the bending stress. Thus in order to obtain a realistic contact pressure, a convergence study on the contact zone is necessary.

- Carrier pin stiffness

Another very important piece of data while considering the epicyclic gear train is the stiffness of the planet carrier pin. As observed on the contact zone, the stiffness of the mounting can affect stress distribution in the region of the engaged teeth. Since the carrier pin stiffness affects the center line of the planet gear, the deflection of the carrier pin changes the contact patch position on the planet gear. From the FEA analysis, the carrier pin stiffness can be obtained by comparing the tangential displacement between two ends of the planet carrier pin over its length. (Fig. 3.12)

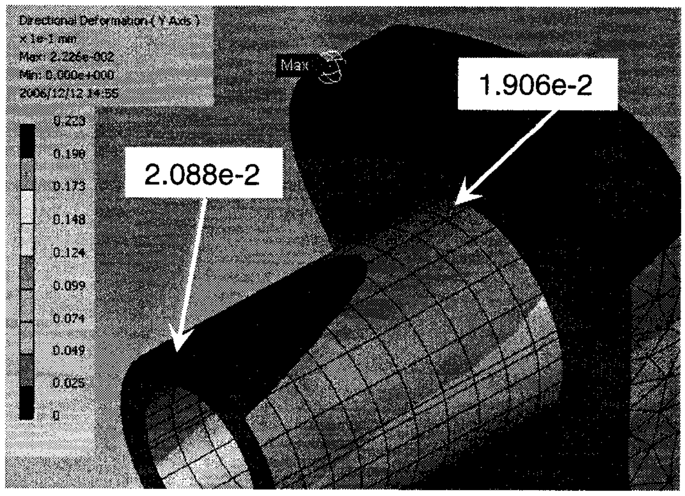


Fig. 3.12 Planet carrier pin tangential displacement

An evaluation of the planet's carrier pin stiffness can be obtained by calculating the relative tangential movement of the two ends of the carrier pin, Eq. (3.1)

$$D' = \frac{D_1 - D_2}{L} \tag{3.1}$$

where D' relative deflection of carrier pin

D_1 maximum tangential displacement on one end of carrier pin

D_2 tangential displacement on the other end of carrier pin

L carrier pin length

In this gear train model, L=150 mm, D_1 and D_2 are obtained from Fig. 3.9, the relative deflection is:

$$D' = \frac{D_1 - D_2}{L} = \frac{2.088\times10^{-2} - 1.906\times10^{-2}}{150} = 0.000012 mm/mm$$

This result is used to evaluate the stiffness of carrier pin, by comparing it with allowable relative deflection, i.e. 0.0006 mm/mm.

3.5 Conclusion

A complete FEA procedure for epicyclic gear trains in the environment ANSYS Workbench has been presented in this chapter. The procedure can be summarized as follows:

- Create three-dimensional geometry model of epicyclic gear train in CATIA V5, then assemble the parts with their relative position;
- Import geometry model into ANSYS Workbench with an associative reader, which keeps the geometry information, material and relative position;
- Simplify and process the geometry model in Design Modeler module, i.e. cut the geometry model to a cyclic symmetry section, Cut-Glue the gear teeth section to control the hexahedral meshing;
- Transfer modified geometry model to Simulation module;
- Mesh the geometry model, refine engaged gear teeth section with appropriate element size for accurate FEA result;
- Define contacts, load and boundary conditions; eliminate conflicts on boundary conditions at interfacing surfaces;
- Launch FEA solution;
- Results analysis. Contact and bending stresses and carrier pin stiffness can be obtained to compare with performance indexes.

This procedure is suitable for the finite element analysis of epicyclic gear train. Although this procedure is tested on a simplified epicyclic gear train, a FE analysis on complete epicyclic gear train can be also achieved with small additional processes on the geometry model. If designers want to launch FEA on the gear train repeatedly with different combinations of geometry dimensions or loads, an automatic solution can be realized by an xml format program that is supported by ANSYS Workbench.

CHAPTER 4

OPTIMIZATION OF EPICYCLIC GEAR TRAIN IN ANSYS WORKBENCH

Optimization of the preliminary design is very important for the mechanical system that has high demands on weight, volume, strength or stiffness. Previous studies on the optimization design of gear system focused mainly on the mathematical formulations and by plotting figures of performance – design parameters on their certain variational ranges. These theoretical optimization methods can achieve satisfying optimization results on the general or typical gear sets. But for a real world gear system, there are many detailed geometric characters that are not considered at the theoretical design stage. Most mechanical systems are primarily designed to meet functional and performance requirements. Then optimization of this preliminary design is performed. Since a preliminary design can be very complex and possess many special characteristics, a theoretical-based optimization method is not adequate. The components of a gear train system interact on each other, which is very difficult to express in mathematical relations. Lots of design parameters in a whole system also increase the complexity of optimization on global system. An optimization procedure on the pre-designed geometry model using a powerful commercial tool can provide a comprehensive and satisfying optimal result. This chapter presents the methodology to perform an optimization design on the epicyclic gear train in the environment of ANSYS Workbench.

4.1 Principal Optimization Methods in ANSYS Workbench

ANSYS is powerful multi-physics FE analysis and optimization software issued by ANSYS INC. ANSYS Workbench is special platform of ANSYS that can directly deal with a three-dimensional geometry model. Its three principal modules, Design Modeler, Simulation and DesignXplorer, can create a 3-D geometry model, perform a FEA solution, and optimize the whole system respectively. DesignXplorer, as an optimization module of ANSYS Workbench, is a powerful tool for designing and understanding the analysis results of mechanical system. Based on Design of Experiments (DOE) technique and various other optimization methods, DesignXplorer uses parameters as its basic language to perform a design optimization. These parameters can come from Design Modeler, Simulation, or other supported CAD system, such as CATIA V5. Structural and thermal responses can be studied, quantified and graphed.

The optimization methods available in DesignXplorer are Design of Experiments (DOE), Variational Technology (VT), Six Sigma Analysis and Robust Design. Within the frame of this project, DOE and VT are considered as the appropriate methods for epicyclic gear train optimization. This chapter discusses the application and optimization procedure of DOE and VT.

4.1.1 Design of Experiments (DOE) vs. Variational Technology (VT)

DOE is a general investigation technique adopted by the finite analysis method to determine the location of sampling points. Sampling points are located such that the space of input parameters is explored in the most efficient way. This efficient location method not only reduces the number of sampling points, but also increases the accuracy of the response surface that is derived from the results on the sampling points. VT provides a much more efficient approach by providing a response surface that is based on a single finite element solve. This technology is combined with the use of mesh morphing and the Taylor series expansion approximation. Since the derivatives are also calculated, this extended finite element analysis needs longer computing time than a regular solve. However, compared to the many solutions run in a regular DOE solve, time consumption in this extended finite element analysis is considerably less. The difference in time consumption between these two technologies is more significant with the increase of design parameter numbers. (Table 4.1)

Table 4.1 Difference of FEA solution runs between DOE & VT

Number of Design Parameters	**Necessary FEA Solution Runs in DOE**	**Solution runs in VT**
2	9	One single extended FEA solution
3	15	
4	25	
5	27	
6	45	
8	81	
10	149	
20	553	

Besides the significant computing time difference between DOE and VT, there are still several other advantages of applying VT on the optimization. These advantages can be compared with DOE as below (Table 4.2):

Table 4.2 Comparison of DOE and VT

DOE	**VT**
● No FE solution displayed on the model	● Full FE solution displayed on the model
● Practical limit of 5 to 20 parameters	● Allows more than 20 parameters
● Moderately accurate	● Highly accurate
● CPU intensive	● Very fast for discrete and material parameters
◊ Very general, suitable for any analysis type	◊ Sensitive to mesh influence, should avoid mesh variation during optimization process. Other limitations are presented in 4.1.2

4.1.2 Limitations in Variational Technology

As seen from comparison of DOE and VT in the previous section, VT surmounts DOE in the computing time, limiting of parameter number, solution accuracy and availability of FE solution display. But with these advantages, VT also has some weaknesses and limitations. One of the weaknesses of VT is that it is sensitive to mesh variation. If the meshing pattern has been changed due to the large dimension variation, VT will stop the optimization process. To ensure a successful VT optimization, the variation range of dimension parameter can not be large and is normally less than ±10% when tested by experiment.

This weakness is not major if compared to other limitations. These limitations may impede the performance of VT on certain optimization target, but it has to be mentioned that these limitations only affect specific optimization problems, as on the epicyclic gear train in the scope of this research.

1. Only bonded MPC contact is supported. In epicyclic gear systems, contact between meshed teeth and contact between the carrier pin and planet gear are frictionless sliding contacts. For accuracy and for properly simulating the real conditions, these contacts should be frictionless contacts.
2. Coordinate Systems are not supported. In the FE analysis of epicyclic gear trains, several modeling techniques need supplementary coordinate systems.
   - One is for applying Match Face Mesh on the two side cuts for cyclic symmetry (described in section 3.4.4).
   - Another is for applying CPCYC command (cyclic symmetry) to couple the nodes on the two cuts of the section model.
   - The third one is for applying CPINTF in order to define the cylindrical joint between the carrier pin and planet gear central hole. Applying CPINTF is a way to conveniently replace the bearing type contact which occurs at the interface between the parts. CPINTF couples only the radial degrees of the freedom between the two groups of nodes respectively.

In two last cases, the coordinate systems can be created by APDL commands, which can successfully take advantage of the limitations mentioned, but the identical mesh required on the cuts (Match Face Mesh) cannot be bypassed because it is a GUI (Graphical User Interface) process and is dealt with before activating the embedded APDL command.

The limitations of VT in the current ANSYS Workbench version make it unreliable for the epicyclic gear train model to perform an accurate FE analysis and optimization. Considering this situation, and in spite of the larger resources involved, DOE is chosen to perform a relatively accurate FE analysis and optimization and VT is used to perform an approximate analysis and optimization by using linear (bonded) contacts. The utilization of these two technologies on epicyclic gear train models is presented in the following sections.

4.2 Optimization by DOE in ANSYS Workbench

With appropriate preparation of the FE analysis in Simulation module; performing a DOE based optimization is medium-complex task. Before running an optimization, all the design parameters and output parameters have to be exposed and conveniently linked in order to assure a successful geometry update during the design space exploration. Design parameters include dimension of geometry model or material properties. Output parameters can be mass or volume, and any results of FE solution as stress, strain or deformation.

For a preliminary study of the epicyclic gear train we chose a limited number of parameters. Within CATIA and ANSYS Design Modeler the input (design) parameters are set. Within ANSYS Simulation and ANSYS DesignXplorer the output parameters are respectively declared. Their definitions as well as their variation ranges are listed in Table 4.3.

Table 4.3 Input & Output Parameters in Epicyclic Gear Train Example

	Parameter Name	Initial Value	Lower Bound	Upper Bound
Design Parameters	Width of Gears	80 mm	72 mm	88 mm
	Radius of Sun Gear Shaft	50 mm	45 mm	55 mm
	Radius of Lighting Holes in Planet Gear	20 mm	18 mm	22 mm
	Width of Carrier	40 mm	36 mm	44 mm
	Radius of Carrier Shaft	100 mm	90 mm	110 mm
Output Parameters	Total Mass			
	Maximum von Mises Stress			

Design parameters listed above are indicated in Fig. 4.1.

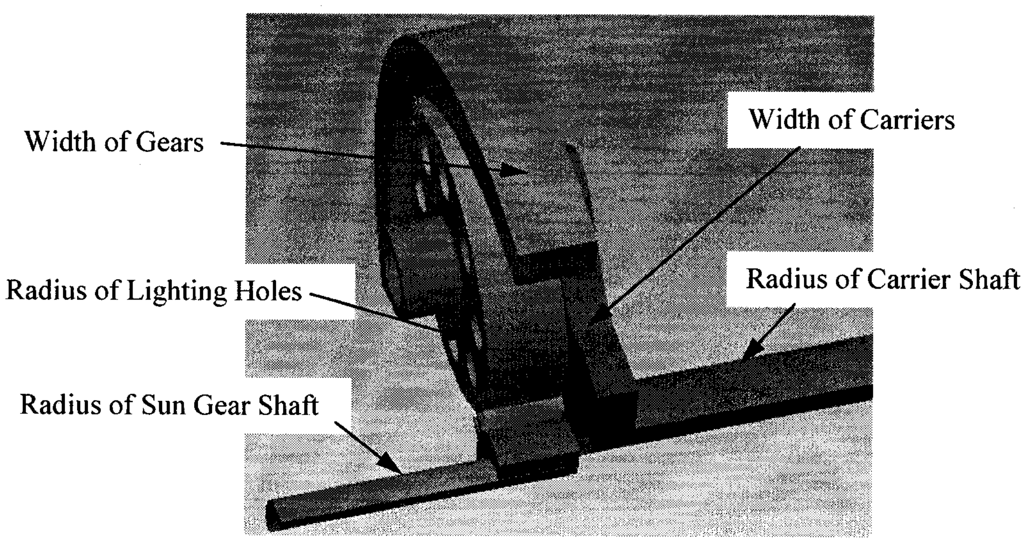


Fig. 4.1 Design Parameters in Epicyclic Gear Train Example

Once all the parameters have been properly defined, the lower and upper limits are set for each design parameter (Table 4.3). If needed, derivative parameters employed as output parameters are defined in this step too. Finally, an optimization run is launched. For the epicyclic gear train model, the total optimization time used by DOE is around six hours on an Intel Pentium D platform with a dual-core 3.0 GHz CPU.

Once the optimization run is successfully executed, the main part of optimization process is finished. Response surfaces are obtained and the sensitivity study can be processed.

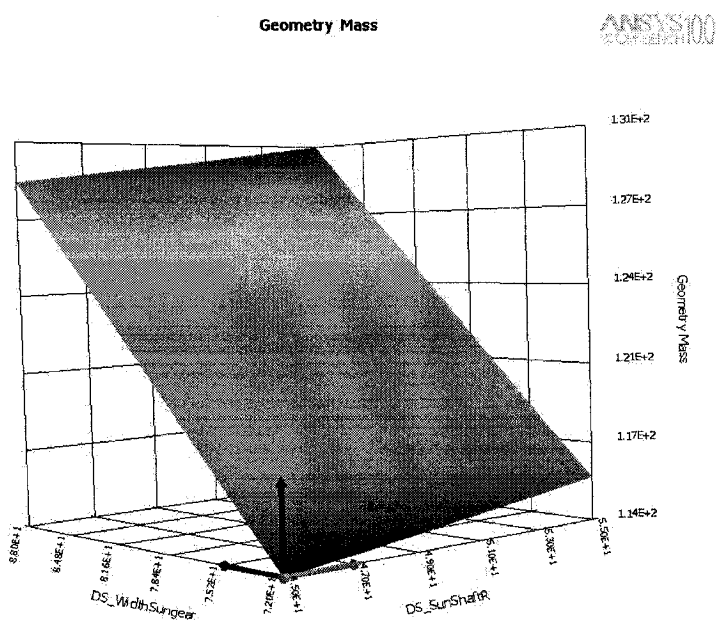
4.3 Parameter Sensitivity Study in ANSYS Workbench

The Parameter Sensitivity study is an important part in the design optimization. A complex mechanical system may have multiple design parameters. Some parameters have a great influence on the system response, whereas others may have less or little influence on the system response. At the first stage of optimization design, putting all the design parameters on one optimization run is not an appropriate strategy. It will prolong the optimization time and may not lead to a satisfactory result. Due to the different magnitudes of influence of design parameters on the system performance, an efficient routine first performs a global and preliminary optimization with large influence design parameters, then a refined and second stage optimization with less influence on design parameters.

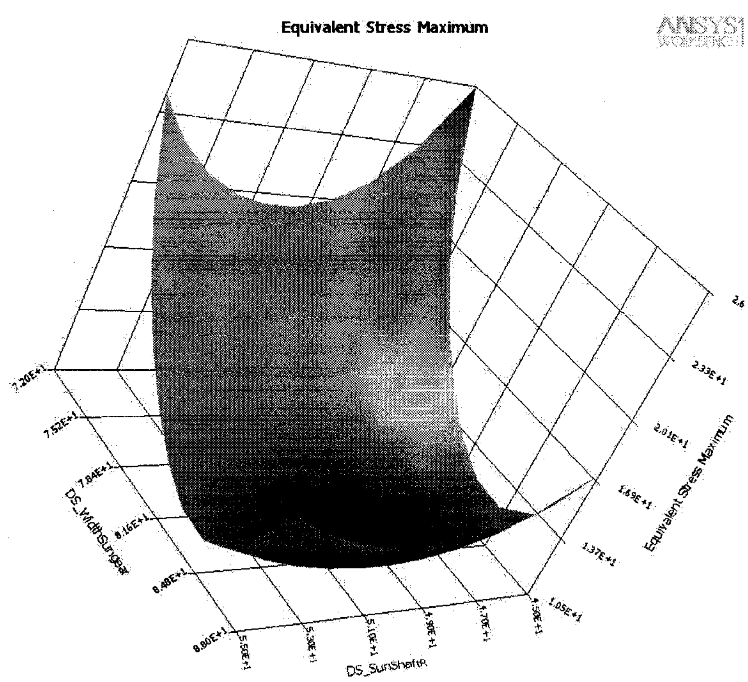
At this stage we can adopt two kinds of sensitivity study methods. The first one comes from issues in RSM (Response Surface Method), which analyzes the influence of each couple of design parameters on a certain response. The second one is in the "Single Parameter Sensitivity Study", which provides a percentage of the influence of all design parameters in a single response. The main benefit of the "Single Parameter Sensitivity Study" is that a global view of influence on all design parameters in a single output parameter can be obtained.

4.3.1 Response Surface Analysis

With "Response Surface", designers can determine the relative influence of each two-design parameters on one output parameter. A response surface from epicyclic gear train optimization in respect to the parameters given in Table 4.3 is shown in Fig. 4.2.



(a) Surface response of dimension parameters to total mass



(b) Surface response of dimension parameters to maximum von Mises stress

Fig. 4.2 Surface response of design parameters on output parameters

Fig. 4.2 (a) shows the variation of the total mass with the change of the gear width and the radius of the sun gear shaft. The figure yields that the gear width has more influence than sun gear shaft radius on the total mass. The relation between the two input parameters and the output parameter is approximately linear. Since the optimization objective is the minimum total mass, we can predict by Fig. 4.2 (a) these two input parameters should be as near to their lower bounds as possible.

In Fig. 4.2 (b), the influence of the same two-design parameters on maximum von Mises stress is shown. This time, the relation between the input parameters and the output parameter is non-linear. For a specific von Mises stress, the values of input parameters are located somewhere between the lower and the upper bounds. As in the Fig. 4.2 (a), the gear width has more influence on the von Mises stress than the radius of sun gear shaft.

4.3.2 Single Parameter Sensitivity Study

Besides the "Surface Response" that studies the influence of one or two design parameters at one time, "Single Parameter Sensitivity Study" provides a complete study of the influence of all design parameters to a specific output parameter. The results of parameter sensitivity study on the epicyclic gear train are shown in Fig. 4.3. Length of the bar shows the degree of influence of each design parameter. Positive value means that with the design parameter value increasing, the response parameter value also increases. Whereas negative value means that with the design parameter value increasing, the response parameter value decreases.

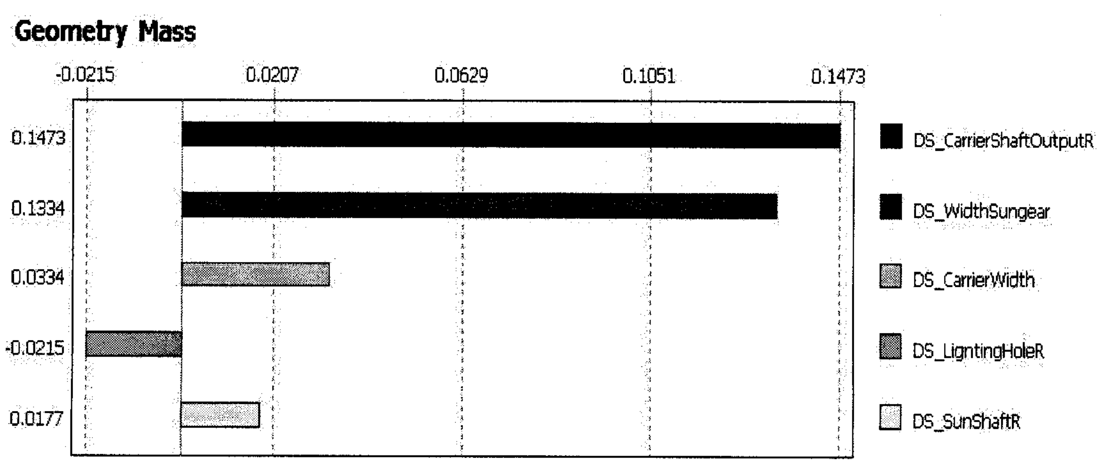


Fig. 4.3 Parameter Sensitivity Study

Sensitivity value can be calculated by Eq.(4.1). With the specific range of the design parameter value (default is ±10%), sensitivity value is equal to the difference of response parameter value dividing the initial response parameter value.

$$Sen_j = \left(\frac{y_{\max} - y_{\min}}{y_{ini}} \right)_j \tag{4.1}$$

where Sen_j single parameter sensitivity for design parameter j;

y_{max} corresponding upper response parameter value for input parameter j;

y_{min} corresponding lower response parameter value for input parameter j;

y_{ini} initial response parameter value.

Besides the bar chart shown in Fig. 4.3, a direct view of parameter sensitivity can be obtained by Pie Chart in the sensitivity study (Fig. 4.4).

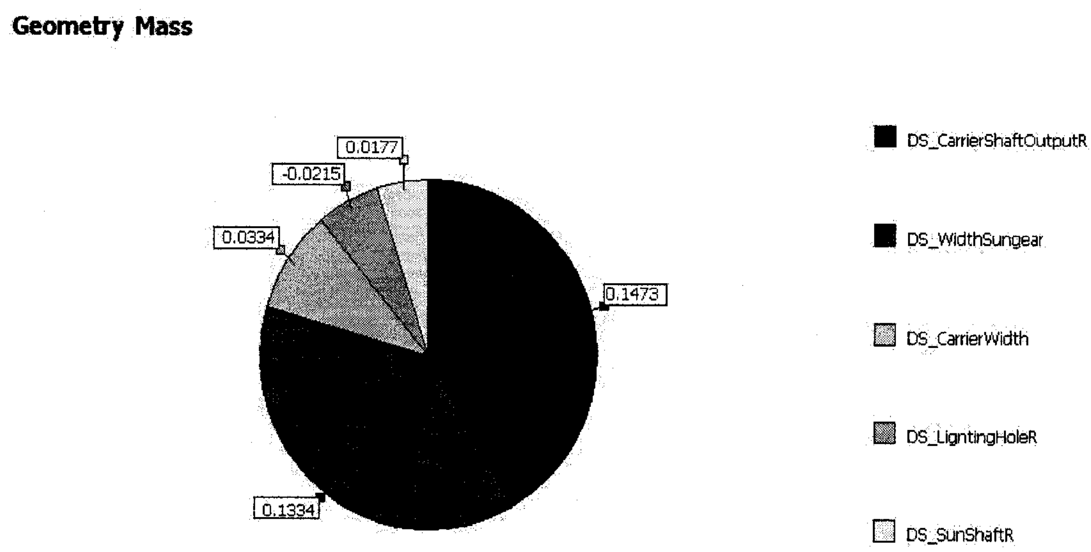


Fig. 4.4 Pie Chart of Parameter Sensitivity

Influence of all the design parameters is quantified by percentage.

$$P_j = \left(\frac{Sen_j}{\sum_{j=1}^{n} Sen_j} \right)_j \tag{4.2}$$

where Sen_j single parameter sensitivity for design parameter j;

P_j sensitivity percentage for design parameter j.

The results of single parameter sensitivity study on epicyclic gear train are shown in Table 4.4.

Table 4.4 Results of sensitivity study on epicyclic gear train

Input Parameters	Variation Range	Parameter Sensitivity	Sensitivity Percentage	Output Parameter
Radius of Carrier Shaft	90-110 mm	0.1473	41.7%	Global mass
Width of Gears	72-88 mm	0.1334	37.8%	
Width of Carrier	36-44 mm	0.0334	9.4%	
Radius of Lighting Holes in Planet Gear	18-22 mm	-0.0215	6.1%	
Radius of Sun Gear Shaft	45-55 mm	0.0177	5.0%	

Table 4.4 shows that the radius of the carrier shaft and the gear width are dominant among the input parameters on specific output parameter – global mass. Then these two parameters should be selected in the preliminary optimization.

The sensitivity study provides a global view on design parameters, which greatly orients designers so that they may properly control the optimization procedure. As mentioned at the beginning of this chapter, two or three phases of the optimization procedure with different combinations of design parameters can provide a quick and global outline of the whole gear train system. After the design parameters have been properly adjusted to approach the final optimal design, a refined optimization on all design parameters can be done simultaneously. After the parameter sensitivity study, a "Goal Driven Optimization" is performed to obtain the best candidates, according to objective functions defined by the designers.

4.4 Goal Driven Optimization

Goal Driven Optimization (GDO) is a constrained, multi-objective optimization (MOO) technique in which the "best" possible designs are obtained from a sample set given by the goals set for the parameters. The sample set is generated either by the "Screening" or "Advanced" method. The "Screening" approach is a non-iterative direct sampling method by a quasi-random number generator, whereas the "Advanced" approach is an iterative Multi-objective Genetic Algorithm (MOGA), which can optimize problem-solving with continuous input parameters. Problems with non-continuous parameters, such as discrete parameters, can not be handled by the "Advanced" approach currently. Generally the "Screening" approach is used for preliminary design while the "Advanced" approach is for refined optimization design. In GDO, certain preferences on the output parameters are defined as the optimal goals, i.e. minimum total mass, as well as maximum value constraints on certain stresses or displacements. Finally, GDO will search for the best candidates from the generated sample set to satisfy these preferences to the greatest possible extent.

The optimization goal on the epicyclic gear train is set as a combination of global mass and maximum von Mises stress. This is the multi-objective optimization. The goal is to minimize the global mass meanwhile the maximum von Mises is kept at the same level. Thus the maximum von Mises is set as of high importance and the global mass is set as of intermediate importance. Generally, DesignXplorer will give the three best candidates for selection. These three candidates match the optimization goal to a different extent (Table 4.5).

Table 4.5 Optimization result of an epicyclic gear train

	Before optimization	After optimization		
		Candidate A	Candidate B	Candidate C
Maximum von Mises (MPa)	11.8	11.9	12.9	12.3
Total mass (kg)	122.5	117.9	115.9	118.0
Reduction of total mass		-3.8%		

Since the optimization goals of minimizing of total mass, and keeping the maximum stress are conflicting goals, the optimization procedure has trade-off characteristics. For example, considering the three candidates in the optimization of a gear train, if maximum stress is more important than total mass, then candidate A is selected, which results in 3.8% reduction of system total mass while the maximum von Mises stress is kept at the same level as before the optimization. This case shows that a lower mass can be achieved but with a higher stress. The final candidate results from a compromise between the former ones.

We may state that further criteria will allow the designer to better orient the design loops in order to reach the optimum.

CHAPTER 5

DEVELOPMENT OF AN OPTIMIZATION METHOD BASED ON GENETIC ALGORITHM (GA)

An optimization process on the epicyclic gear train geometry model by DesignXplorer module of ANSYS Workbench was presented in CHAPTER 4. Both Design of Experiment (DOE) and Variational Technology (VT) are employed in the optimization respectively. As a large commercial FEA tool, ANSYS Workbench provides a complete FEA solution package which includes geometry modeling, FEA meshing and solution, and optimization. Their benefits allow designers to design, analyze and optimize a mechanical system in a very convenient way. But this does not mean that the commercial tool is appropriate in all cases. The highly integrated commercial tool is designed mainly to process the most general mechanical system, and normally many configuration settings have been predefined to reduce the manual interventions. For the complex or specific mechanical system, a commercial tool is not guaranteed to solve the analysis and optimization problems directly, or in a way that designers expected; and some predefined settings can not be modified conveniently by users. Thus, a controllable and setting-changeable optimization module is usually demanded for in the analysis and optimization procedure.

The second main part of this research task is to develop a FE based optimization procedure. We choose to develop a GA code for optimization of the gear train. A GA based optimization method is presented in CHAPTER 5. In CHAPTER 6, specific GA optimizers are developed both on the C++ environment and the MATLAB environment. An integration of the developed optimization module and a commercial FE analysis tool is proposed at the end of CHAPTER 6, which allows the FE analysis and GA based optimization to be executed automatically.

5.1 Principles of Genetic Algorithms

Genetic Algorithm (GA) is a heuristic search algorithm based on Darwin's nature selection theory. Basic GA theory in the optimization design will translate the design variables into binary code, and evaluate the performance of the mechanical system as to the fitness of selection. Performance evaluations are usually the objective of optimization. The candidates with a higher fitness are kept as the parents to generate the next generation. As the process loops, fitness is increased more and more which means the optimization objective is approached gradually.

5.2 Application of GA on Engineering Optimization

An engineering optimizer is developed with a general Genetic Algorithm strategy (Goldberg, 1989). The optimization procedure is shown below:

1. Create the optimization model, define design variables;
2. Set variable boundaries, the generation number and the population number;
3. Generate the first generation of candidates randomly (each candidate is a set of design variables);
4. Decode binary code of design variables into real value;
5. Send each candidate to the analysis module to run a solution; obtain the results for each candidate;
6. Calculate the fitness of each candidate by the objective function;
7. Sort the candidates with the fitness value;
8. Check the stop criteria, if meet, stop the optimization, keep the best candidate as the optimization result;
9. Select half of the candidates; those who have better fitness as the "parents" for the next generation;
10. Convert fitness into chance of selection (better fitness has higher chance to be selected);

11. Using the "roulette" method (nature selection theory) generate the next generation, candidates with better fitness have more chance to be selected to the next generation. Crossover and mutation occur depending on their predefined occurrence possibilities;
12. Repeat step 4 to 11, until the stop criteria are met.
13. Optimization process is terminated. The best candidate in the last generation is kept as the final optimization result.

5.3 Multi-objective function

This developed GA optimizer can deal with both single objective optimization and multi-objective optimization. Single objective optimization is relatively easy since the optimization objective is treated directly as the fitness of selection. But for multi-objective optimization, it is quite complex since the optimal design is not a single point but a set of trade-off study points, which come from the conflicting optimization goals, i.e. total volume and maximum stress in a mechanical system.

For the preliminary stage of optimization, a basic multi-objective optimization method is applied within the scope of this research. Considering a mechanical system as the optimization object, total volume and maximum stress are the objectives of optimization. The optimization directive is that total volume has to be reduced as much as possible and maximum stress must be as close as possible to the allowable stress. Since reduced volume comes from the geometry's small dimension, which will increase the maximum stress, the two objectives conflict each other. To apply a basic multi-objective optimization method, a fitness function is written as Eq.(5.1),

$$f = V + a_1 \cdot \log(s_1 - S_1)^2 + a_2 \cdot \log(s_2 - S_2)^2 + \cdots a_n \cdot \log(s_n - S_n)^2 \qquad (5.1)$$

where f fitness function or objective function

V total volume

$s_1, s_2, \ldots, s_n$	optimization objectives, i.e. maximum stresses, maximum deformation
$S_1, S_2, \ldots, S_N$	allowable limits of optimization goal for each objective
$a_1, a_2, \ldots, a_n$	weighting factors for optimization objectives

The effect of weighting factors $a_1, a_2, \ldots, a_n$ is to balance the weight of each objective, or to make them comparable. If the weighting factors are not carefully studied and chosen, the variations of some objectives may not affect the global fitness. Then the targets of some objectives may never be reached.

A logarithm function is used as a penalty function in Eq.(5.1). During the optimization, if the result of an output parameter (optimization objective) is far from its target value, fitness of this objective will be considerably increased. Thus the total fitness will also be increased. Then the optimizer will push this objective to approach its target value to reduce the total fitness. (For the discussed model in Chapter 6, the better candidates have smaller fitness.)

This balanced fitness function tries to search for a combination of design variables in order to meet all the requirements of objectives. This can be realized by adjusting the weighting factor for each optimization objective. The more important objective has a higher weighting factor value. Thus little changes on the more important objective will significantly change the total fitness. On the other hand, the total fitness is not sensitive to the changes in objectives with lower weighting factor values. At the end of the optimization procedure, the best fitness comes from a combination of all the optimization objectives, but with different weighting factors.

CHAPTER 6

DEVELOPMENT OF SPECIFIC GA OPTIMIZERS AND INTEGRATION WITH DIFFERENT ANALYSIS TOOLS

In the previous chapter, a genetic algorithm optimization procedure was presented and a multi-objective function was created. In this chapter, two GA optimizers are developed, one in the C++ language environment and the other in the MATLAB language environment. C++ language is a powerful programming language, thus it is suitable for building a complex optimization system with graphic user interface. Whereas MATLAB is more concise and flexible, it is convenient for engineering computing.

6.1 Development of GA Optimizer in C++ Environment and the Application on Analytical Model

A GA optimizer was developed in C++ environment using a genetic algorithm optimization procedure discussed in section 5.2. A one-stage gear train model was developed to verify the optimization procedure of the GA module. Since our purpose is to test the optimization procedure, the gearbox model should be simple to save computing time and to properly test the methodology. Once the developed optimizer is validated, the optimizer can be applied on a complex mechanical system.

The model is a one-stage gearbox with two gears and two shafts (Fig. 6.1). To simplify the calculation, absolute maximum von Mises stress on the model is considered as the objective of optimization. Another objective is the total volume of the gearbox. The two gears are treated as two cylinders but the force is transmitted between the gears in respect to gear drive theory. An input torque T_1 is applied on one end of pinion shaft and one end of gear shaft is fixed. Both ends of shafts are supported simply. Load and support conditions are shown in Fig. 6.1.

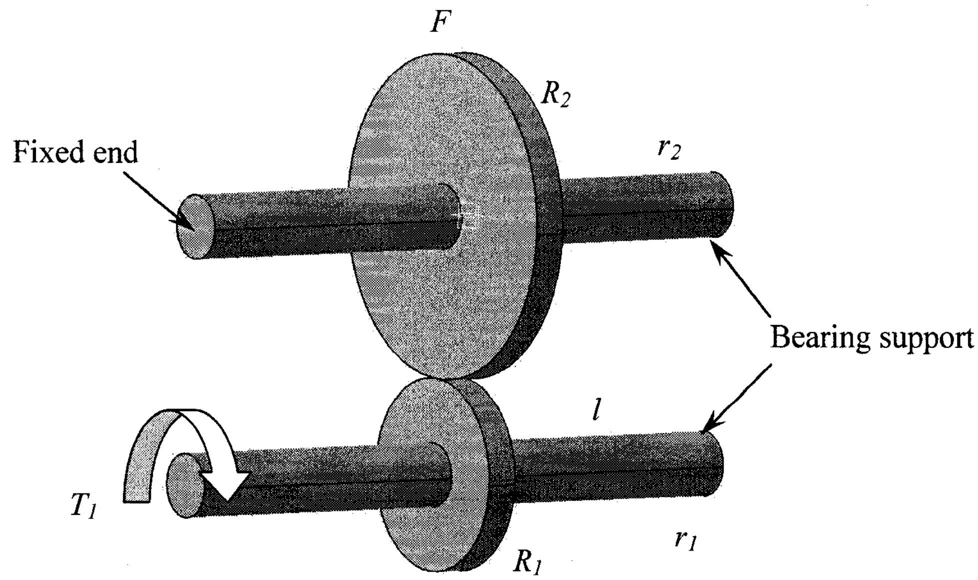


Fig. 6.1 One stage gearbox

Assuming r_1 and r_2 are the radius of the pinion shaft and the radius of the gear shaft respectively, R_1 and R_2 are the radius of the pinion and the radius of the gear respectively.

Tangential force F_t on pinion teeth is: (F_t is also the tangential force on gear teeth)

$$F_t = \frac{T_1}{R_1} \tag{6.1}$$

Radial force between two gears is:

$$F_r = F_t \times tg\phi \tag{6.2}$$

where ϕ is the pressure angle.

The total normal contact force F_{all} on the tooth profile is:

$$F_{all} = \sqrt{F_t^2 + F_r^2} \tag{6.3}$$

From Eq. (6.1) to Eq. (6.3),

$$F_{all} = \frac{T_1}{R_1}\sqrt{1 + tg^2\phi} \tag{6.4}$$

The bending moment M on the center of shaft is:

$$M = \frac{1}{2} F_{all} \times \frac{1}{2} l = \frac{1}{4} \cdot \frac{T_1 l}{R_1} \sqrt{1 + tg^2 \phi} \tag{6.5}$$

where l is the length of shaft between the supports.

Maximum bending stress σ and maximum torsional stress τ can be obtained by Eq. (6.6) and Eq. (6.7), respectively,

$$\sigma = \frac{32M}{\pi d^3} \tag{6.6}$$

$$\tau_{xy} = \frac{16T}{\pi d^3} \tag{6.7}$$

where d is the diameter of the shaft.

The maximum von Mises stress σ_{equiv} is obtained by Eq. (6.8),

$$\sigma_{equiv} = \sqrt{\sigma^2 + 3\tau_{xy}^2} \tag{6.8}$$

The maximum von Mises stress in pinion shaft $\sigma_{equiv.1}$ can be derived from Eq. (6.5) to Eq. (6.8),

$$\sigma_{equiv.1} = \frac{T_1}{\pi r_1^3} \sqrt{\frac{l^2}{R_1^2} \left(1 + tg^2 \phi\right) + 12} \tag{6.9}$$

and the maximum von Mises stress in gear shaft $\sigma_{equiv.2}$ is:

$$\sigma_{equiv.2} = \frac{T_2}{\pi r_2^3} \sqrt{\frac{l^2}{R_2^2} \left(1 + tg^2 \phi\right) + 12} \tag{6.10}$$

Since

$$\frac{T_2}{R_2} = \frac{T_1}{R_1}$$

Then Eq. (6.10) can be written as:

$$\sigma_{equiv.2} = \frac{T_1}{\pi r_2^3 R_1} \sqrt{l^2 \left(1 + tg^2 \phi\right) + 12 R_2^2} \tag{6.11}$$

The total volume V of this gearbox can be approximately calculated by:

$$V = \pi \left(r_1^2 + r_2^2\right) l + \pi \left[\left(R_1^2 - r_1^2\right) + \left(R_2^2 - r_2^2\right)\right] F \tag{6.12}$$

where F is the width of the two gears.

Since all the equations are established, r_1, r_2, R_1, R_2 are defined as design variables. T_1, l F and ϕ are constant and are pre-defined as:

Input torque: $T_1 = 100000$ Nmm   Gear width: $F = 10$ mm

Shaft length: $l = 200$ mm   Pressure angle: $\phi = 20^\circ$

The fitness function is a multi-objective function which is a combination of total volume, $\sigma_{equiv.1}$ and $\sigma_{equiv.2}$. By giving lower and upper bounds for each design variable, an optimization on the analytical model is performed. Here we set 30 as the population number in each generation and maximum generation is set to 50. Design variables and results are shown in Table. 6.1.

Table 6.1 Design variables and optimization results

Design & Output Variables	**Bounds or target values**	**Optimization results**
Radius of pinion R_1	30-40 mm	34.92 mm
Radius of gear R_2	60-80 mm	69.96 mm
Radius of pinion shaft r_1	5-15 mm	7.74 mm
Radius of gear shaft r_2	5-15 mm	6.75 mm
Total volume V	minimum	258339
Max. pinion shaft stress $\sigma_{equiv.1}$	480 MPa	479.98 MPa
Max. gear shaft stress $\sigma_{equiv.2}$	480 Mpa	477.88 MPa

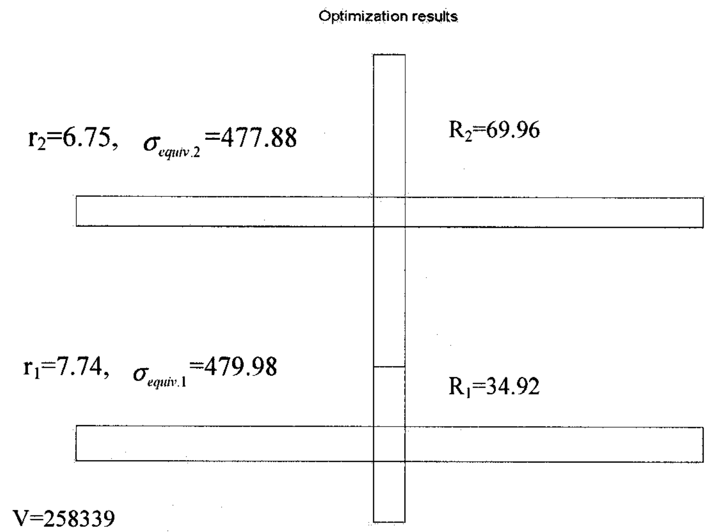


Fig. 6.2 Optimization result with C++ based optimizer GA on analytical model

Fig. 6.2 shows the optimization results. In this model, the direction of optimization can be predicted before launching the optimizer. Decreasing the radii of pinion shaft and gear shaft can reduce the total volume, but will increase the maximum stresses. Thus there is a critical point where the total volume is at the minimum while maximum stresses do not exceed allowable stress (480Mpa in this example). The point found by the optimizer (Fig. 6.2) is a good design point because the maximum stresses are near their target values.

6.2 Development of GA Optimizer in MATLAB Environment

Since C++ is a powerful programming language, it can build complex optimization systems like commercial software, including graphic user interface, and menu tools bars. C++ is more suitable for advanced program development.

MATLAB is a high-level technical computing language and it provides an interactive environment for algorithm development, data visualization, data analysis, and numeric computation. It integrates computation, visualization, and programming in an easy-to-use environment where problems and solutions are expressed in familiar mathematical notation. In MATLAB, basic elements such as variables and matrix do not require definition or dimensioning before they are used. Their powerful function library makes technical computation more convenient. Thus in the scope of this research, MATLAB language as a strong engineering computing tool has been selected to build a preliminary GA optimizer.

A GA based optimizer is developed using MATLAB programming language. Design variables are encoded in binary code. The optimization target for the mechanical system was articulated within an objective function containing maximum stress and/or total volume. Selection fitness combines multi-objective function with different weighting factors. Crossover and mutation occur depending upon their predefined occurrence possibilities. Population and the number of generations are modified to make sure that the optimal design can be found efficiently, since a too-large population or generation considerably increases the computing time without improving accuracy.
The GA optimizer is applied on both an analytical model and a finite element analysis model of the same one-stage gearbox. An analytical model can provide a preliminary solution for the mechanical system. The FEA method can solve complex mechanical problems with a more precise solution, which makes it a popular solution method in modern mechanical design. Once this developed GA optimizer is validated on both of the two models, and result in a similar optimal design, this GA optimizer can be directly applied onto the optimization of complex mechanical systems.

6.2.1 Application of MATLAB based GA Optimizer on Analytical Model

The analytical model to be optimized is still a one-stage gearbox. This gearbox is composed of two spur gears and two shafts. To simplify the model, there is only one pair of teeth meshed at one time. This gearbox model is shown in Fig. 6.3. Unlike the gearbox model studied in section 6.1, where maximum stresses on the two shafts are considered, in this chapter gear teeth stress such as the maximum bending stress is calculated and selected as the optimization objective.

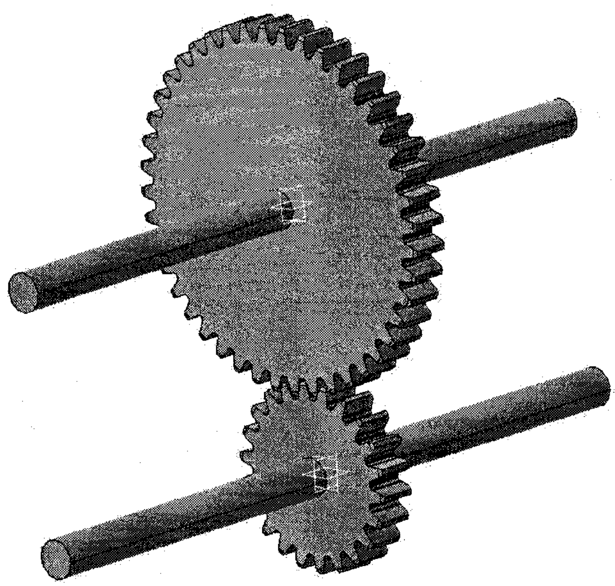


Fig. 6.3 One stage gearbox with meshed teeth

<u>Gear Tooth Bending Stress</u>

The first equation developed for gear tooth bending stress is the Lewis equation. This equation is derived by considering the tooth as a simple cantilever, with tooth contact occurring at the tip as shown in Fig. 6.4. The equation was announced in 1892 and it still stands as the basis of gear design today.

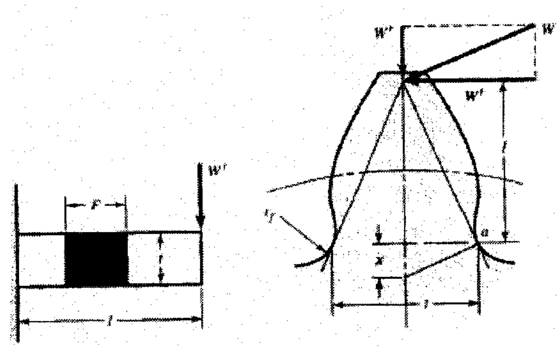


Fig. 6.4 Lewis gear tooth bending stress model

It is assumed that only one pair of teeth is in contact and only the tangent component (W_t) will be considered. To derive the basic Lewis equation, Fig. 6.4 shows a cantilever of cross-sectional dimensions F and t, having a length l and a tangential load W_t uniformly distributed across the distance F. The maximum bending stress in a gear tooth occurs at point a. Using the standard equation for bending stress ($\sigma = Mc/I$), the gear tooth bending stress is:

$$\sigma_b = \frac{Mc}{I} = \frac{6W_t l}{Ft^2} \tag{6.13}$$

where I is the moment of inertia;

c is the distance from neutral surface to outer surface.

By similar triangles, $\frac{t/2}{x} = \frac{l}{t/2}$, then $x = \frac{t^2}{4l}$,

Now substitute the value of x into Eq. (6.13) and multiple the numerators by the circular pitch p, then

$$\sigma_b = \frac{W_t p}{F(2/3)xp}$$

Letting $y = 2x/3p$, we get the original Lewis equation:

$$\sigma_b = \frac{W_t}{Fpy} \tag{6.14}$$

The factor y is called Lewis form factor. In practice, most engineers prefer to use the diametral pitch P in determining the tooth bending stress. By substituting $P = \pi / p$ and $Y = \pi y$ in Eq. (6.14), a more usual form of Lewis equation is shown as:

$$\sigma_b = \frac{W_t P}{FY} \tag{6.15}$$

Y is Lewis form factor and can be obtained from most machine design textbooks.

Optimization on Analytical Model

The gear tooth bending stress derived in the previous section is settled upon as one optimization objective. Another objective is the total volume of the gearbox. Since less total volume tends to increase the bending stress, the optimization is a trade-off optimization procedure. A multi-objective optimization method that is discussed in section 5.2.2 is applied to this analytical model.

The total volume is calculated by Eq. (6.12),

$$V = \pi\left(r_1^2 + r_2^2\right)l + \pi\left[\left(R_1^2 - r_1^2\right) + \left(R_2^2 - r_2^2\right)\right]F$$

where r_1 radius of pinion shaft,

r_2 radius of gear shaft,

R_1 radius of pinion pitch circle,

R_2 radius of gear pitch circle,

F face width.

Then the objective function is:

$$f = V + a_1 \cdot \log\left(\sigma_b - S\right)^2 \tag{6.16}$$

where S is the target value of σ_b.

Dimensions of gearbox are defined as design variables. Their lower and upper bounds are shown in Table 6.2:

Table 6.2 Bounds of design variables for GA optimization

Design Variables	**Lower Bound (mm)**	**Upper Bound (mm)**
R1	25	40
R2	60	80
r1	10	20
r2	10	20

Other constant values are assumed as below:

Input torque T = 20000 Nmm

Face width F = 10 mm

Pressure angle $\phi = 20^\circ$

Gear modulus m = 4 mm

The function of weighting factor a_1 is to balance the different optimization objectives and to make them comparable. Since in this example the value of total volume is around 3×10^5 and value of tooth bending stress is around 60, thus weighting factor a_1 is defined as 5×10^3.

Since the analytical model is described by mathematical formulas, optimization is all defined in the MATLAB environment. The convergence evolution of the optimization process is shown in Fig. 6.5.

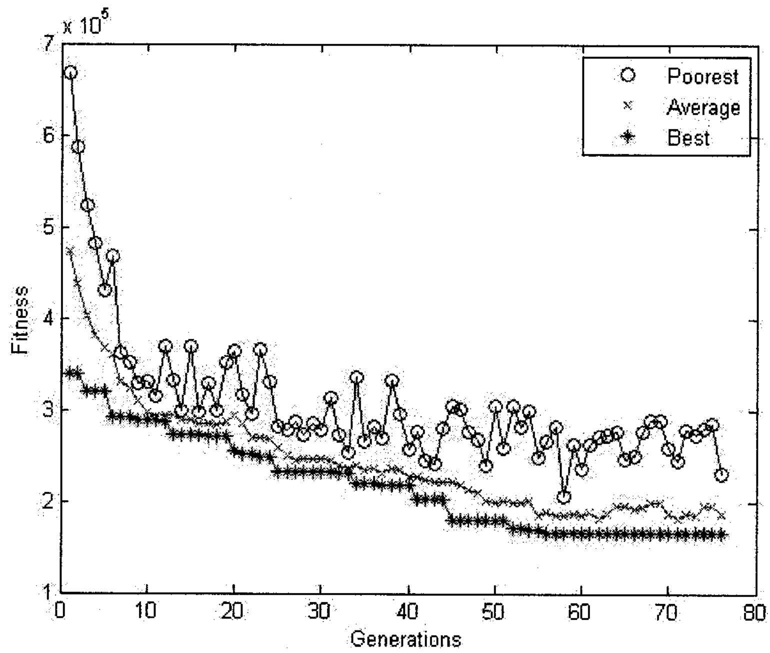


Fig. 6.5 Optimization process on analytical model

Optimization process is stopped at the 76th generation. The processing time is around 10 seconds in an Intel dual-core 3.0GHz platform. A stop criterion is defined as Eq.(6.17) :

$$Stop\_criterion = \frac{f_{i-20} - f_i}{f_{i-20}} \times 100\% \leq 0.8\% \quad (i \geq 30) \tag{6.17}$$

where f_i best fitness in the i^{th} generation;

f_{i-20} best fitness in the $(i-20)^{th}$ generation;

Starting from the 30th generation, the best multi-objective fitness is compared with the best fitness in the 20th last generation. If the best fitness does not change significantly (less than 0.8%) during 20 generations (see Fig. 6.5), we can say the optimization is converged; the process can be stopped. Values of design variables in the last generation (76th generation in this example) are kept as the optimization result. (See Table 6.3):

Table 6.3 Optimization results on analytical model

Parameters	**Bounds or target values**	**Optimization results**
Radius of pinion R_1	25-40 mm	30.19 mm
Radius of gear R_2	60-80 mm	60.31 mm
Radius of pinion shaft r_1	10-20 mm	10.07 mm
Radius of gear shaft r_2	10-20 mm	10.00 mm
Multi-objective fitness f	minimum possible	165470
Total volume V	minimum possible	241636 mm^3
Tooth bending stress σ_b	60 MPa	59.999 MPa

According to Eq.(6.15), the tooth bending stress is only affected by tangential force W_t, which is affected by radius of pinion R_1 for the constant input torque. To obtain a minimal total volume, the other three design variables r_1, r_2 and R_2 should be as small as possible. From optimization result in Table 6.3, these three variables are near the lower bounds of their limits, and the tooth bending stress is very close to its target value. Then we can conclude that the optimization successfully found the optimal result for this analytical model.

6.2.2 Application of MATLAB based GA Optimizer on FEA Model

In the previous section, a one-stage spur gearbox is analyzed and optimized using an analytical formulation. The same gearbox will be analyzed in a FEA model and be optimized by the same optimizer to compare the optimization procedure and their results. FE analysis is executed in ANSYS environment and is called by the GA optimizer to process the optimization.

6.2.2.1 Create and Analyze the same Optimization Model in ANSYS

A one-stage gearbox model (two gears and two shafts) of the same dimensions is created in ANSYS. In order to parameterize the design variables and to be able to update the geometry model automatically, the gearbox model is created by a developed ANSYS APDL (ANSYS Parametric Design Language) program. Other processes, such as meshing model, applying loads and boundary conditions, running a solution and gathering the results, are also included in the same APDL program. This program can be executed either by ANSYS in a macro mode or by an external Windows command in batch mode.

The geometric model of a gearbox with meshing elements and boundary conditions is shown in Fig. 6.6. The FEA model is built with a bottom-up procedure. Gear cylinder and shafts are meshed with a coarse element, whereas engaged teeth are meshed with a refined element to increase the accuracy.

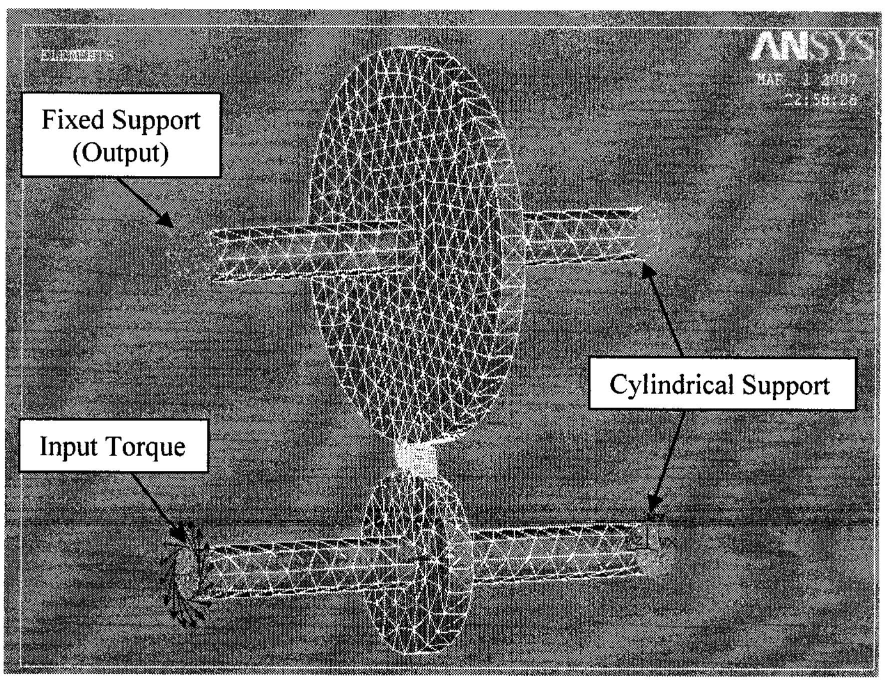


Fig. 6.6 Meshed gearbox model with boundary conditions

The whole procedure of finite element analysis, from creating the model to exporting analysis results is accomplished automatically with a single APDL program. The procedure can be summarized as below:

1. Read gearbox dimension from an input file (input.txt);
2. Create pinion, gear and two shafts;
3. Define element size for teeth (refined) and other parts (coarse), mesh whole model;
4. Apply load (input torque) and boundary conditions (fixed support and cylindrical support);
5. Define contact between two meshed teeth faces;
6. Perform simulation;
7. Calculate the total volume and post-process the maximum tooth bending stress;
8. Export two results into an output file (result.txt).

6.2.2.2 Optimization by Integration of GA Optimizer and ANSYS

The optimization procedure based on the FEA model is similar to the one which was applied to the analytical model. The main difference is that analytical model is described by a MATLAB program and lies in the same environment with GA optimizer, whereas the FEA model is created, and simulation is performed in the ANSYS environment. The communication between ANSYS and MATLAB is the key point for automatically executing the optimization run.

ANSYS provides a batch mode which allows the running of the ANSYS simulation task in the background by a Windows command. Thus the optimization can be realized automatically by a batch command. This optimization procedure can be described by Fig. 6.7.

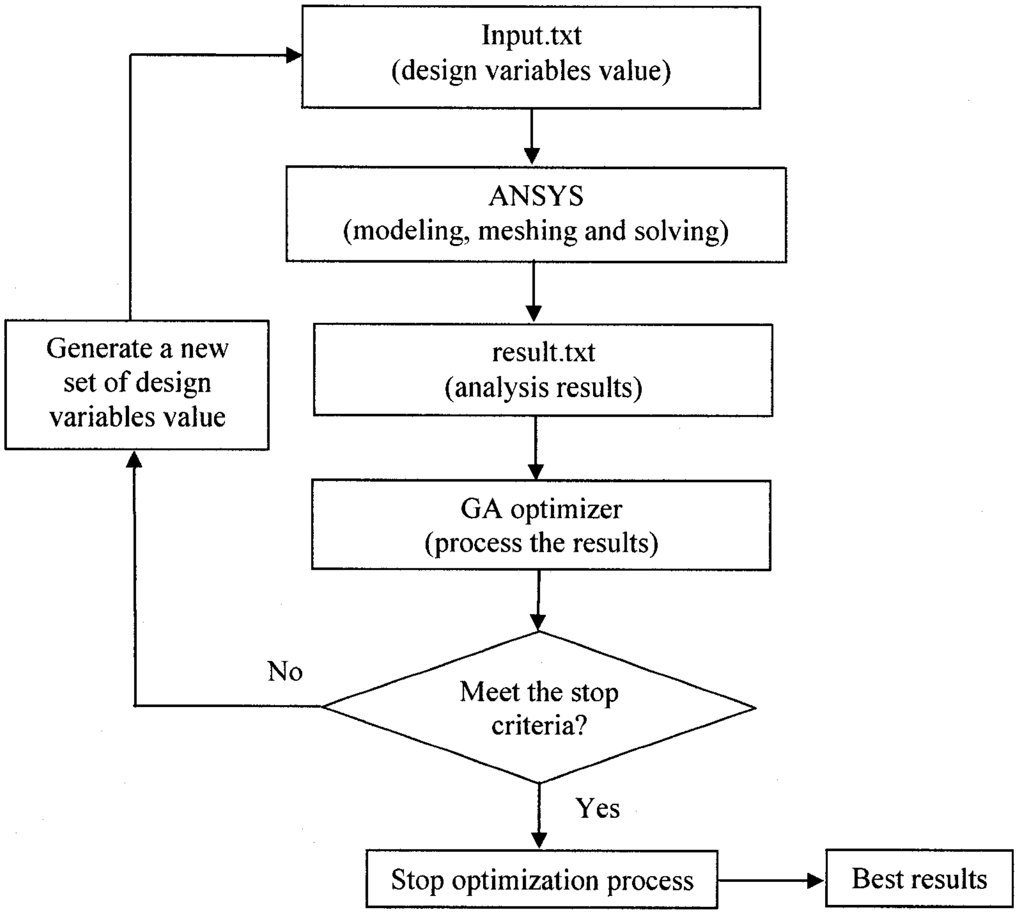


Fig. 6.7 Optimization procedure on FEA model

The command calls ANSYS to run a simulation task, and then exports calculation results into an output file. GA optimizer (a MATLAB code program) reads the calculation results, evaluates the results with the stop criteria, and generates a new set of variables to launch the next ANSYS simulation. This procedure will continue until the predefined number of generations is executed, or the "stop" criterion is met, before the total number of generations can be executed. Then the best candidate is extracted as the final optimal design.

Fig. 6.7 also shows an integration procedure with the different analysis and optimization tools. The analysis tool is a FEA application running in ANSYS environment and the optimization tool is a GA optimizer running in MATLAB environment. With this integration method, different applications are called by the integrated system to execute their assigned task in a same Windows platform. Therefore an automatic optimization procedure is achieved. The same methodology can also be used on the integration of the developed GA optimizer with other analysis tools.

GA optimization on the FEA model is realized using the same conditions as those used on the analytical model. The optimization process is shown in Fig. 6.8. The optimization trend on the FEA model is also similar to that of the analytical model. To simplify the FEA solution and to save analysis time, the teeth profile in the FEA model is not involute but circular. Furthermore, to prevent a considerably long non-linear solution, the contact type on meshed teeth is defined as bonded contact. This is the reason why bending stress is considered in our problem but not contact stress.

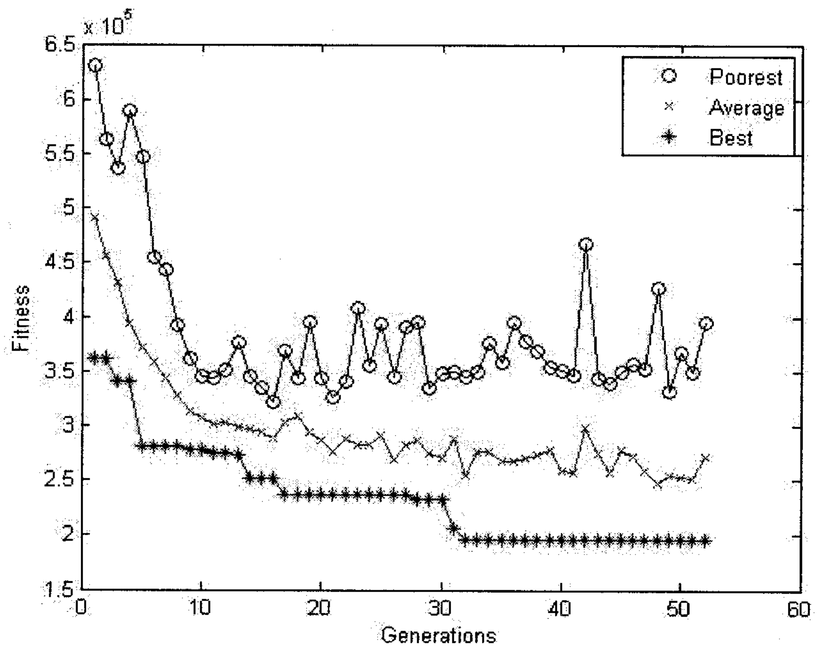


Fig. 6.8 Optimization process on FEA model

The optimization process on the FEA model is stopped at the 52nd generation. The total processing time is around 4.5 hours (in the same platform: Intel dual-core 3.0GHz). Optimization results from the last generation are shown in Table 6.4.

Table 6.4 Optimization results on FEA model

Parameters	Bounds or target values	Optimization results
Radius of pinion R_1	25-40 mm	37.80 mm
Radius of gear R_2	60-80 mm	61.27 mm
Radius of pinion shaft r_1	10-20 mm	12.93 mm
Radius of gear shaft r_2	10-20 mm	11.06 mm
Multi-objective fitness f	minimum possible	201393
Total volume V	minimum possible	307157 mm^3
Tooth bending stress σ_b	60 MPa	59.999 MPa

From the analysis of optimization results on the analytical model, we found that in this gearbox model the tooth bending stress is only related to the radius of the pinion. So to obtain a minimum total volume, the value of three other variables should be as little as possible. Tooth bending stress should be as close to its target value as possible. From the optimization results shown in Table 6.4, these goals are all reached by the GA optimizer. Optimization on the FEA model in ANSYS environment is successfully fulfilled by the Genetic Algorithm optimizer.

6.2.2.3 Precision on the Number of Population and Generation

Convergence speed during a GA optimization is always an issue. It highly depends upon the definition of the analysis model and the selection of design variables. If design variables do not have similar parameter sensitivities to the analysis model, convergence of optimization will be slow, since it takes time for the GA optimizer to find the optimal design. Theoretically, optimal design can be found by increasing the population number and/or generation number in the optimization process. But large population numbers and generation numbers dramatically increase optimization time, especially for an optimization based on a complex finite element analysis model.

To deal with this problem, the optimization model should be carefully analyzed by designers before running the optimization, and design variables should be selected properly. Furthermore, a "stop" criterion can prevent optimization from wasting a great deal of time if no more progress is to be accomplished during the optimization process. These two strategies are applied in the GA optimization with both the analytical model and the FEA model presented in this chapter.

6.3 Comparison of Optimization based on Different Analysis Tools

The same one-stage gearbox model is optimized in both the analytical model and the FEA model found in sections 6.2.1 and 6.2.2. To evaluate the performance of our GA optimizer on the two models, optimization results are compared in Table 6.5. The results show that optimization objectives are reached and that all the constraints have been respected. Minimum total volume has been approached and the target value of bending stress has been reached. In both models, the optimum value of pinion radius has been found to make tooth bending stress close to its target value. The other three variables are near their lower bounds because they do not influence tooth bending stress but only affect the total volume.

Table 6.5 Comparison of optimization results on analytical model and on FEA model

Parameters	**Bounds or target values**	**Optimization Results (analytical model)**	**Optimization Results (FEA model)**
Radius of pinion R_1	25-40 mm	30.19 mm	37.80 mm
Radius of gear R_2	60-80 mm	60.03 mm	61.27 mm
Radius of pinion shaft r_1	10-20 mm	10.07 mm	12.93 mm
Radius of gear shaft r_2	10-20 mm	10.00 mm	11.06 mm
Multi-objective fitness f	minimum	165470	201393
Total volume V	minimum	241636 mm^3	307157 mm^3
Tooth bending stress σ_b	60 MPa	59.999 MPa	59.999 MPa
Generation number when optimization stopped		76th	52nd
Total processing time		10 seconds	4.5 hours

The developed GA optimizer has been validated for both the analytical model and the FEA model of the same gearbox, and optimization results are very close. Therefore, though it is very difficult for a complex gear train model to be described by an analytical model, it can be optimized successfully by this GA optimizer which is based on a FEA method. Since an optimization based on a FEA method takes a significantly long processing time, a preliminary optimization employing the analytical method is important, and a parameter sensitivity study is very helpful. Finally, a precise FEA - based optimization is performed to obtain, or to validate the final design.

CHAPTER 7

CONCLUSION

7.1. Original Contribution

A general finite element analysis procedure on an epicyclic gear train was developed in this thesis. This FEA procedure was specified within ANSYS Workbench environment. The whole gear train model was reduced and simplified in order to acquire accurate and convergent solutions. The contact and bending stress observed were in accordance with classic gear theory.

The contact status proves a correct simulation of frictionless contact between the meshed teeth. Teeth surface contact stress and teeth bending stress are obtained. The contact pressure is distributed on a contact line between the meshed gears. The algorithm used, as well as the surface to surface definition of the contact, allowed us to obtain a realistic distribution of the contact pressure. For the ring – planet meshed gears the contact position was centered on the contact line, but for the sun-planet couple we found that maximum pressure was concentrated on one end of the contact line. One can say that the modeling was accurate and precious information was collected regarding the design of parts (shafts, web design) where the gears are installed. In conclusion, an inappropriate distribution of supporting parts stiffness may affect the contact patch in terms of distribution and location.

The convergence study on the sun – planet contact zone (the highest contact stress) showed the contact pressure converged during the decreasing of element size, whereas the bending stress was stable. The contact pressure is more sensitive than the bending stress in relation to the element size. Thus, in order to obtain a realistic contact pressure, a convergence study always needs to be conducted on the contact zone.

This analysis procedure can be applied to the general epicyclic gear train, and helps gear train designers fulfill a full set of analysis by FEA method.

In the second part, an optimization procedure for epicyclic gear trains was developed in the ANSYS Workbench DesignXplorer environment. The optimization procedure is based on the Design of Experiments techniques. A Response Surface Analysis presented the influence of any two design parameters on a selected output parameter. A Single Parameter Sensitivity Study showed the weighted influence of all design parameters on one output parameter. The same epicyclic gear train model was then optimized to validate this procedure. We studied the change of the total mass with the variation of the gear width and the radius of the sun gear shaft. We observed that the gear width has more influence than sun gear shaft radius on the total mass while the variation is approximately linear. Since the optimization objective is the minimum total mass, we believe that these two input parameters should be as close to their lower bounds as possible. Nevertheless, the influence of the same two design parameters on maximum von Mises stress showed that the relationship between the input parameters and the output parameter is non-linear. As exposed previously, the gear width has more influence on the von Mises stress than the radius of sun gear shaft.

The "Single Parameter Sensitivity Study" provides a complete study of the influence of all design parameters on a specific output parameter. Results emphasize that the radius of the carrier shaft and gear widths are dominant among the input parameters on the specific output parameter – global mass. Accordingly, these two parameters should be selected in the preliminary optimization. The sensitivity study provides a global view on the design parameters, which greatly helps in orienting designers so that they can properly control the optimization procedure.

In applying a Goal Driven Optimization (GDO) which is a multi-objective optimization (MOO) technique, we were able to find the "best" possible design from a sample set.

The multi-objective optimization on the epicyclic gear train was set as a combination of global mass and maximum von Mises. The optimization yielded 3.8% reduction of system total mass while the maximum von Mises stress is kept at the same level as before the optimization. Based on this procedure and by considering further criteria one may be able to better orient the designer in order to reach the optimum configuration

In order to perform a faster preliminary optimization on the epicyclic gear train, optimization modules were developed based on Genetic Algorithms (GA). The optimization procedure searches for the "best" design candidates by simulating the nature selection process. A multi-objective function was developed to deal with the conflicting design variables. Penalty function was employed to push the design variables close to their target values.

Two GA based optimizers were developed, one in the MATLAB environment and the other in C++ environment. The GA optimizer in C++ code was applied on a one-stage gearbox to validate the optimization procedure. The results show that the optimal design variable values were found to meet the optimization targets.

The last part proposed a MATLAB tool, based on GA integrated with both the analytical and the finite element analysis formulations. The convergence figures show that optimal results can be reached in an efficient way. The GA optimizer was validated on a simple gearbox model, and thus ensured; it was applied onto the complex mechanical system optimization.

By comparing the different techniques and types of integration, we identified that for the desired design procedure which would be as fast and accurate as possible in the integrated use an optimizer as early as possible is the most appropriate. This integration based on an analytical formulation of the mechanical system during the preliminary phase of design, allows a faster approach to the best configuration in a time efficient way. Meanwhile, Finite Elements Analysis sensitivity studies generally provide the information needed to build the best formulation of the objective function used by the optimizer. Whereas the analytical formulation is based only on norms and standards (here AGMA standards), an advanced Finite Elements based solution provides the complete contribution of different parts to the general and the local behavior of the mechanical system. This final step of the design procedure is destined then to validate the global optimum solution.

7.2. Future Research Discussion

The research proposed by the present work was the preliminary stage of finite element analysis and optimization of an epicyclic gear train. The geometric model is simplified to perform a global finite element analysis on the whole system. The contacts between meshed teeth are defined with surface to surface contact using an Augmented Lagrange formulation. Even if an almost zero penetration is achieved, further assessments and convergence studies must be performed. The contact definition and the post-processing procedure should be one of future research works, especially if variational technology is chosen as optimization tool.

Variational Technology (VT) as a highly efficient technique should be also considered. Due to its limitations on sliding contact definition and coordinate systems, research is needed to find a feasible way to profit from the advantages of VT, especially when a large number of parameters have to be considered. The high sensitivity of VT to the mesh morphing makes VT more suitable for the local refined optimization method.

In order to create an automatic design, and reduce the product cycle development, the GA optimizer can be developed so that it integrates CAD tools (such as CATIA) with CAE tools (such as ANSYS Workbench). This should be realized by further developing the GA optimizer in the powerful C++ programming environment. Once the interfaces between different tools are well established, and the data exchange rules are properly defined, an automatic gear train design system is accomplished. Then the main modules of this system, such as the analytical design, the 3D geometry modeling, the FEA simulation and the GA optimization, can be called and performed automatically. The routine is controlled and repeated by the program until the best optimal design point is found.

REFERENCE

Brauer, J., (2004), A general finite element model of involute gears, Finite Elements in Analysis and Design 40 (2004) 1857-1872

Bush, G. S., Osman, M. O. M., Sankar, S., (1984), ON THE OPTIMAL DESIGN OF MULTI-SPEED GEAR TRAINS, Mechanism and Machine Theory, Vol. 19, No. 2, pp. 183-195

Carroll, R. K., Johnson, G. E., (1989), Dimensionless Solution to the Optimal Design of Spur Gear Sets, ASME, J. of Mech., Trans., and Auto. in Des., Vol. 111, pp. 290-296

Cavdar, K., Karpat, F., Babalik, F. C., (2005), Computer Aided Analysis of Bending Strength of Involute Spur Gears with Asymmetric Profile, J. of Mech. Des. MAY 2005, Vol. 127 pp 477-484

Chong, T. H., Kubo, A., (1985), Simple stress formulas for a thin rimmed spur gear, ASME, J. Meca., Transm., Autom. Des., 107, pp. 406-423

Deb. K., Jain, S., (2003), Multi-Speed Gearbox Design Using Multi-Objective Evolutionary Algorithms, J. of Mech. Des., Sept. 2003, Vol. 125, pp. 609-619

Documentation of ANSYS, version 10.0

Documentation of ANSYS Workbench, version 10.0

Goldberg, D. E., (1989), Genetic Algorithm in Search, Optimization and Machine Learning. Addison-Wesley

Jain, P., Agogino, A. M., (1990), Theory of Design: An Optimization Perspective, Mech. Mach. Theory, 25(3), pp. 287-303

Li, C. H., Chiou, H. S., Hung, C., Chang, Y. Y., Yen, C. C., (2002), Integration of finite element analysis and optimum design on gear systems, FE in Analysis and Design 38 (2002) 179-192.

Liu, C. C., Tsay, C. B., (2001), Mathematical models of beveloid gears generated by taper hobbing and infeed grinding, J. Chinese Soc. Mech. Eng. 22 (3) (2001) 177-186

Oda, S., Nagamura, K,. and Aoki, K., (1981), Stress analysis of thin rim spur gears by finite element method, Bull. JSME, 24, pp. 1273- 1280

Ramamurti, V., Vijayendra, N.H., and Sujatha, C., (1998), Static and Dynamic Analysis of Spur and Bevel Gears Using FEM, Mech. Mach. Theory, 33, pp.1177-1193

Saravanan, R., Ramakrishnan, K., Surender, S., Dwivedi, A., (2001), Practical Gear Design Optimization using Genetic Algorithm and Simulated Annealing Algorithm, IE(I) Journal-MC, Vol. 82, July 2001, pp. 74-79

Savage, M., Coy, J. J., and Townsend, D. P., (1982), Optimal Tooth Numbers for Compact Standard Spur Gear Sets, ASME Journal of Mech. Des., Vol. 104, Oct., pp. 749-758

Savage, M., Lattime, S. B., Kimmel, J. A., Coe, H. H., (1994), Optimal Design of Compact Spur Gear Reductions, ASME, J. of Mech. Des., Vol, 116, 690-696

Su, D., Qin, D., (2003), Integration of numerical analysis, virtual simulation and finite element analysis for the optimum design of worm gearing, J. of Material Processing Technology 138 (2003) 429-435.

Thompson, D. F., Gupta, S., Shukla, A., (2000), Tradeoff analysis in minimum volume design of multi-stage spur gear reduction units, Mechanism and Machine Theory, 35, pp. 609-627

Tian, Y. T., Li, C. X., Tong, W., and Wu, C. H., (2003), A Finite-Element-Based Study of the Load Distribution of a Heavily Loaded Spur Gear System With Effects of Transmission Shafts and Gear Blanks, J. of Mech. Des., SEPTEMBER 2003, Vol. 125, pp. 625-631

Wang, H., Wang, H. P., (1994), Optimal Engineering Design of Spur Gear Sets, Mech. Mach. Theory, Vol. 29, No. 7, pp. 1071-1080

Wilcox, L; Coleman, W. (1995), Application of finite elements to analysis of gear tooth stresses, ASME J. Eng. Ind., 95, pp.1139-1148

Yokota, T., Taguchi, T., Gen, M., (1998), A Solution Method for Optimal Weight Design Problem of the Gear Using Genetic Algorithms, Computers ind. Engng. Vol. 35, Nos 3-4, pp. 523-526.

ANNEX 1

1.1 Optimization by VT in ANSYS Workbench

Due to inherent limitations in VT technique (discussed in section 4.1.2), an epicyclic gear train can not be performed on a VT optimization currently, with the FEA status of frictionless sliding contact and cyclic symmetry section. But this does not mean that we give up the attempt to profit from the great advantages of VT. This section presents how we can best try to perform an optimization on an epicyclic gear train with the VT technique. Once the limitations of VT have been overcome in the future version of ANSYS Workbench, these experiences will provide valuable assistance to VT-based optimization.

1.1.1 Optimization with VT Technique

Optimizing epicyclic gear train by VT technique is a study in bypassing the limitations of VT.

<u>Bonded Contact Only:</u>

An ANSYS APDL code can partly solve this problem. On the frictionless contact between the planet carrier pin and the planet gear inner hole, an APDL program can simulate the frictionless sliding contact and not incur the definition of frictionless contact in this position. The core of this program is the CPINTF (ANSYS) command, which couples the nodes on the two cylinder surfaces, and frees the Degree of Freedom (DOF) on the tangential direction. But for the sliding contact between the meshed teeth faces, this method can not be applied to the contact teeth faces because the two faces do not contact on the whole face. For VT, a compromise has to be made that these contacts are defined with bonded MPC contacts.

Coordinate System is not supported:

Another APDL code is built to create coordinate systems as needed. This meets the requirement of coordinate systems in coupling the cyclic symmetry faces and coupling the carrier pin contact faces. But for the "Match Face Mesh", this method does not work. This is because the "Match Face Mesh" is always executed before the coordinate system is created by the APDL code. Thus, "Match Face Mesh" has to be deactivated in the meshing process.

With these above compromises, an epicyclic gear train model is ready for a VT optimization in ANSYS Workbench. Like the procedure done in DOE (section 4.2), VT is launched to import the FEA model into the VT optimization module. After the lower and upper limits of each design parameters are properly defined, optimization run can be launched.

1.1.2 Prospective Optimization with New Version of ANSYS Workbench

A DOE based accurate optimization and a VT based approximative optimization are both processed for an epicyclic gear train model in the previous sections. As mentioned before, the DOE method permits the simulation of real epicyclic gear system status at the most extent. DOE is a general and basic optimization method, which does not create a high demand on the preparation of a finite element model. The VT technique may only perform an approximative simulation on the epicyclic gear train due to its insurmountable limitations in its current version, but VT's great advantages always attract designers to profit from its strong functions for mechanical engineering. Once the limitations of DesignXplorer (ANSYS Workbench) (which currently prevent the optimization of epicyclic gear trains) are surmounted in the near future, the more efficient VT technique will considerably improve optimization speed on optimal design.

ANNEX 2

2.1 ANSYS APDL command block for coupling two cuts face of cyclically symmetry model (CPCYC)

```
/PREP7                          ! Enter pre-process module;
CSYS,12                         ! Create global cylindrical coordinate system;
CMSEL,s,sys,node                ! Select nodes on all the two cut faces;
CMSEL,u,edges_sunshaft,node     ! Unselect nodes incurring non-converge problem;
CPCYC,all,0.001,12,,120,,0,     ! Define cyclical symmetry mode;
ALLSEL,all                      ! Select all nodes on the model;
/SOLU                           ! Exit pre-process module, enter simulation module.
```

2.2 ANSYS APDL command block for coupling the nodes on the contact interface (CPINTF)

```
/PREP7                          ! Enter pre-process module;
CSYS,13                         ! Create local cylindrical coordinate system;
CMSEL,s,alesage,node            ! Select nodes on the hole surface (planet gear);
CMSEL,a,arbre,node              ! Select nodes on the shaft surface (carrier pin);
NROTAT,all                      ! Rotate all nodes in accordance with the cylindrical
                                  coordinate system;
CPINTF,ux,0.5                   ! Define the coupling tolerance;
ALLSEL,all                      ! Select all nodes on the model;
/SOLU                           ! Exit pre-process module, enter simulation module.
```